Abstracts XV ISABE
from the
Fifteenth International Symposium
on **Air Breathing Engines**

Edited by Paul J. Waltrup

INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

The ISOABE is proud to acknowledge World Sponsorship by
AIAA, AAAF, CASI, DGLR, RUAeS, RASS
and
US DoD, US NASA and US ONRIFO, Tokyo

Sponsorship in India
Aeronautical Research and Development Board
Defense Research and Development Organization
Aeronautical Development Agency
Hindustan Aeronautics, Ltd.
Council for Scientific and Industrial Research
Aeronautical Society of India
Indian Space Research Organization
Indian Institute of Science Industries

3–7 September 2001
Bangalore, India

REPORT DOCUMENTATION PAGE

Form Approved OMB No. 0704-0188

1. AGENCY USE ONLY (Leave blank)	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	September 2001	September 3-7 2001 Final

4. TITLE AND SUBTITLE
Fifteenth International Symposium on Air Breathing Engines (XV ISABE) Held in Bangalore, India on September 3-7, 2001. Abstracts and Supplementary Book of Abstracts.

5. FUNDING NUMBERS
N68171-01-M-5527

6. AUTHOR(S)
Paul Waltrup, Editor

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
American Institute of Aeronautics and Astronautics
1801 Alexander Bell Drive
Suite 500
Reston, VA 20191-4344

8. PERFORMING ORGANIZATION REPORT NUMBER
ISBN 1-56347-515-4

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)
Office of Naval Research,
European Office
PSC 802 Box 39
FPO AE 09499-0039

10. SPONSORING/MONITORING AGENCY REPORT NUMBER

11. SUPPLEMENTARY NOTES
See also ADM 001333, XV ISABE Proceedings (CD-ROM).
This work relates to Department of the Navy Grant N68171-01-M-5527 issued by the Office of Naval Research International Field Office. The United States has a royalty free license throughout the world in all copyrightable material contained herein.

12a. DISTRIBUTION/AVAILABILITY STATEMENT
Approved for Public Release; Distribution Unlimited.
U.S. Government Rights License. All other rights reserved by the copyright holder.

12b. DISTRIBUTION CODE
A

12. ABSTRACT (Maximum 200 words)
The 15th International Symposium on Air Breathing Engines (XV ISABE) was held at Bangalore, India on September 2-7, 2001, organized by the Indian National Organizing Committee under the sponsorship of various professional organizations across the world, various industries and academic institutions in India, and the governmental authorities in India. The symposium was also supported by various agencies of the US government including the Department of Defense and the National Aeronautics and Space Administration.

13. SUBJECT TERMS
XV ISABE, ONRIFO, Foreign reports

15. NUMBER OF PAGES

16. PRICE CODE

17. SECURITY CLASSIFICATION OF REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19, SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL

NSN 7540-01-280-5500

Standard Form 298 (Rev. 2-89)
Prescribed by ANSI Std. 239-18
298-102

Fifteenth International Symposium
on Air Breathing Engines

SYMPOSIUM ABSTRACTS
3-7 September 2001
Bangalore, India

Edited by
Paul J. Waltrup

Published and distributed by
American Institute of Aeronautics and Astronautics
1801 Alexander Bell Drive, Suite 500
Reston, VA 20191-4344

ISBN 1-56347-515-4

The 2001 ISABE Proceedings will be distributed by the American Institute of Aeronautics and Astronautics, Inc.,
1801 Alexander Bell Drive, Suite 500, Reston, VA 20191-4344
ISABE Proceedings from previous conferences are also available from AIAA.

A Collection of Papers from the 15th International Symposium on Air Breathing Engines Bangalore, India 3-7 September 2001

Table of Contents

FOREWARD

The Fifteenth International Symposium on Airbreathing Engines (XV ISABE) is being held at Bangalore, India, during September 2-7, 2001, organized by the Indian National Organizing Committee, under the sponsorship of various professional organizations across the world, various industries and academic institutions in India, and the governmental authorities in India. The symposium is also supported by various agencies of the US government including the Department of Defense and the National Aeronautics and Space Administration.

The ISOABE continues to be supported by the Member Nations and the Foundation Members. The ISOABE has now become incorporated in the US as a non-profit organization.

The XV ISABE includes an Inaugural Session, the Awards Ceremony, nine plenary sessions with as many invited lectures, and the Closing Ceremony, along with 45 planned contributed paper sessions, including two Forum Sessions.

The Symposium includes the Wu Chung Hua Lecture in Turbomachinery, and the Frederick S. Billig Lecture in Hypersonics.

The Symposium supports two joint sessions, one with the International Astronautical Federation, and the other with the International Council of Aeronautical Sciences. The ISOABE thus provides a unique forum for those interested in airbreathing engines in the field of aerospace activities, along with the other two international organizations.

The Proceedings are being published in CD-ROM format, through the agency of the AIAA; the ISOABE is grateful to the AIAA for this service.

On behalf of the ISOABE Executive Board, the Sponsors in India and in the various Member Nations, the Indian National Organizing Committee, and the City of Bangalore, we extend a warm welcome to everyone to share and assess the developments in Airbreathing Engine Sciences and Technologies and to express the hope that you will enjoy and cherish the activities of the Symposium.

We are deeply appreciative of your support.

S. Michael Hudson
President

K.V.L. Rao
Co-Chair

V. Sundararajan
Co-Chair

Indian National Organizing Committee
June 1, 2001

NATIONS AND ORGANIZATIONS THAT SUPPORT THE INTERNATIONAL SOCIETY OF AIR BREATHING ENGINES

FOUNDATION MEMBERS

- United Technologies: Pratt & Whitney USA
- Von Karman Institute for Fluid Dynamics, Belgium
- Aerospatiales-Missiles, France
- Snecma, Centre de Villaroche, France
- Volvo-Flygmotor AB, Sweden
- General Electric Aircraft Engines, USA
- Rolls Royce plc, UK
- IHI - Ishikawajima-Harima Industries Co., Ltd. Japan
- MTU - Motoren-und Turbinen-Union Munchen GmbH, Germany

NATION MEMBERS

Australia
Aeronautical Research Laboratories, Australian Department of Defense
The University of Queensland
The University of Tasmania

Belgium
Von Karmon Institute for Fluid Dynamics, Rhode St. Genese

Canada
National Research Council of Canada
Canada Aeronautics and Space Institute

China
Society of Engineering Thermophysics, Beijing

Czech Republic
Vyzkummy A Zkusebni Letecky Ustav Letnany, Prague
WALTER Engines

European Community

Egypt
Ain Shams University, Cairo

France
Association Aeronautique et Astronautique de France

Germany
DASA Daimler ChrysleR, Aerospace AG
Deutsches Gesellschaft fur Luft-und Raumfahrt (DGLR)
DLR
MBB-Messerschmitt-Bolkow- Blohm GMBH
MTU-Motoren-und Turbinen-Union Munchen GmbH

Greece
Egniko Ldpyma Epeynon, Ashna
Hellenic Aerospace Industry Ltd.
National Technical University of Athens

India
Aeronautical Research and Development Board, New Delhi
National Aerospace Laboratory, Bangalore

Israel
Beith Shemesh Engine Co.

Italy
Associazoine Temotecnica Italiana, Pisa
University di Florence
University (degli) Studi di Pisa

Japan
Ishikawajima-Harima Heavy Industries, Co., Ltd., Tokyo
The Japan Society for Aeronautical and Space Science
Kyosho University

Korea
Korean Aerospace Research Institute

Netherlands
Technische Universiteit Delft

Norway
A/S Kongsberg Vapenfabrik

Portugal
Instituto Superior Tecnico

Russia
Central Institute of Aviation Motors

Slovak Republic
Slowak Technical University

South Africa
Defence Aeronautics Program CSIR

Spain
IBERIA Lineas Aeras de Espana, Madrid

Sweden
Volvo Flygmotor AB, Trollhattan

Switzerland
Swiss Society of Aeronautical Sciences

Turkey
Istanbul Technical University
Middle Eastern Technical University
Science and Technology Research Council of Turkey
TEI Turkish Industries, Inc.

Ukraine
Ukrainian Academy of Sciences

United Kingdom
Richardo Aerospace
Rolls Royce plc
Royal Aeronautical Society
University of Sheffield

United States
Air Force Systems Command
American Institute of Aeronautics and Astronautics
Boeing Airplane Company
EOARD
General Electric
GE Aircraft Engines
International Astronautical Federation
International Council of Aeronautical Sciences
Lockheed Corporation
McDonnell Douglas Corporation
National Aeronautics and Space Administration
National Science Foundation
Naval Warfare Center
Office of Naval Research
Pratt and Whitney
Rolls-Royce Allison
Sverdrup Technology, Inc.
United Technology

Executive Committee of National Representatives

XV ISABE

Fifteenth International Symposium on Air Breathing Engines

ISABE

INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

September 3, 2001 Bangalore India

Address by the President

S. Michael Hudson

VSM-0373

Figure 1

Simplified Product Model

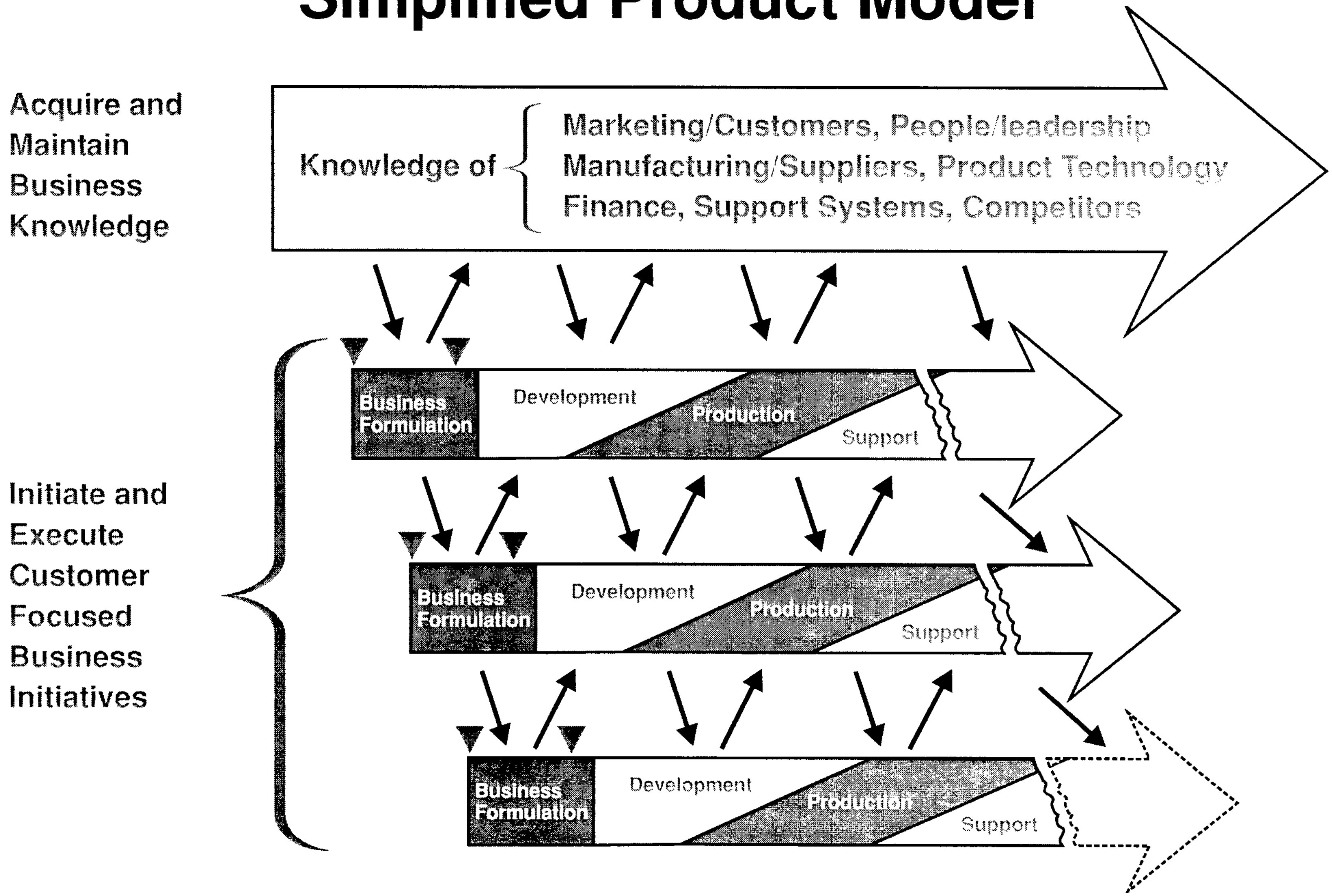
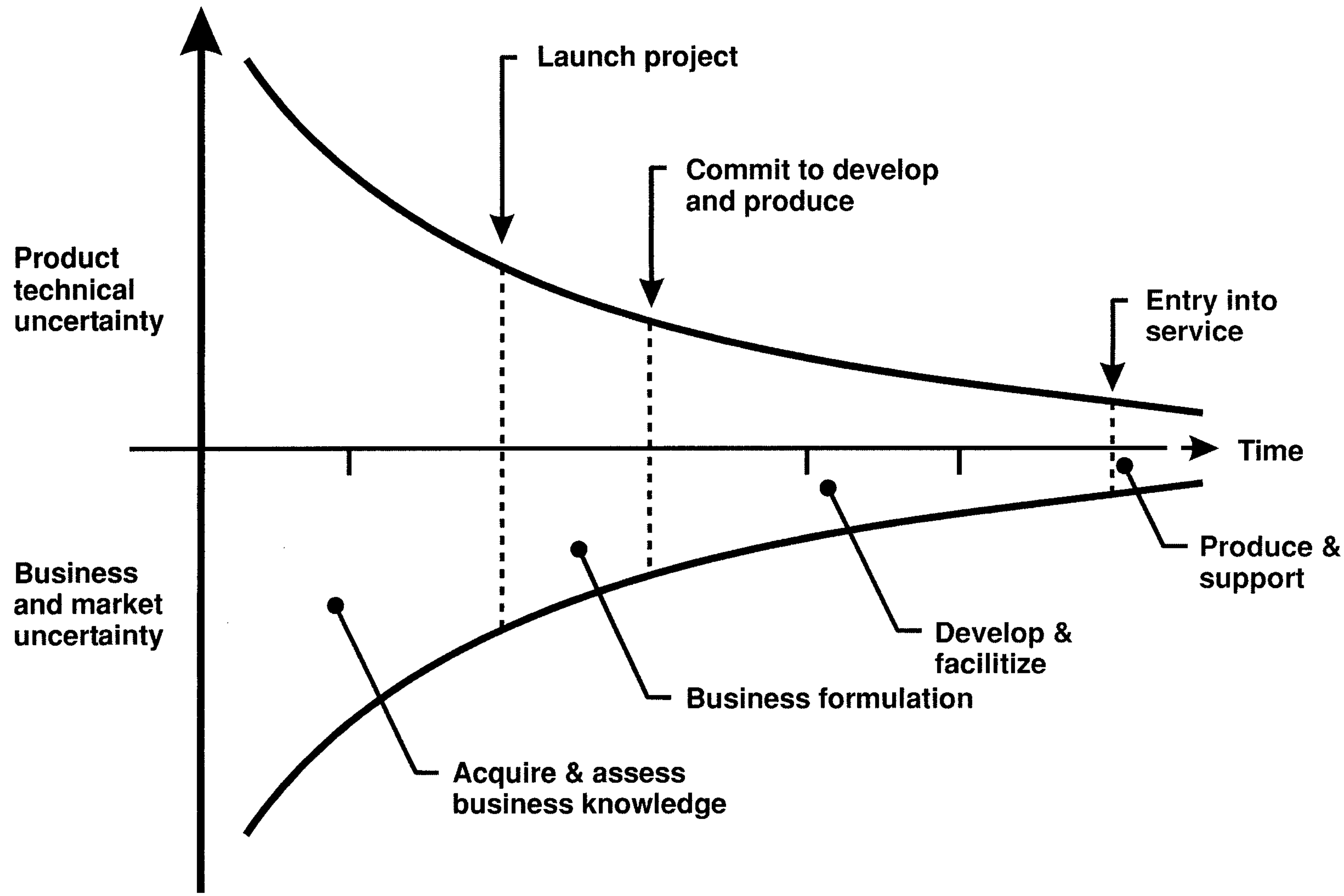


Figure 2

Management of Uncertainty From Formulation Through Project Realization



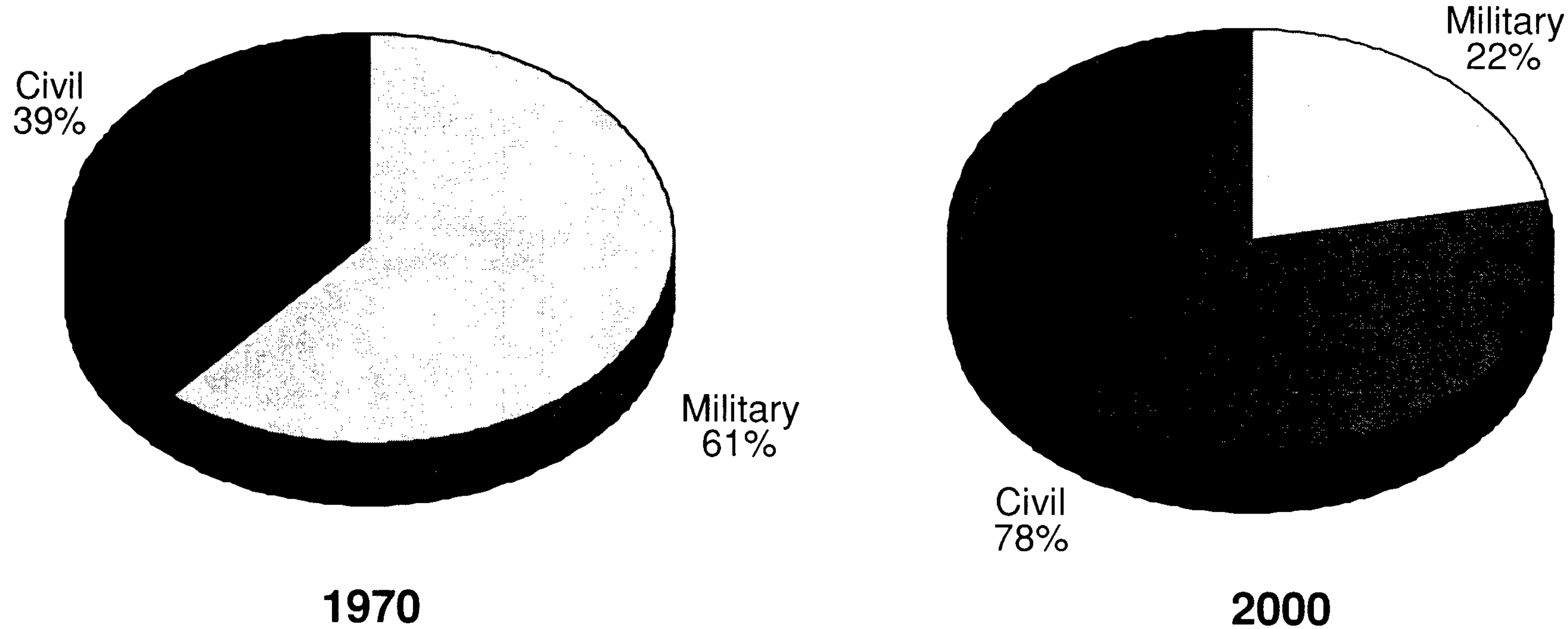
INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

VSM-0377

Figure 3

Government Interest in Air Breathing Engines

Changing Applications and Uses



Specific Government Interest

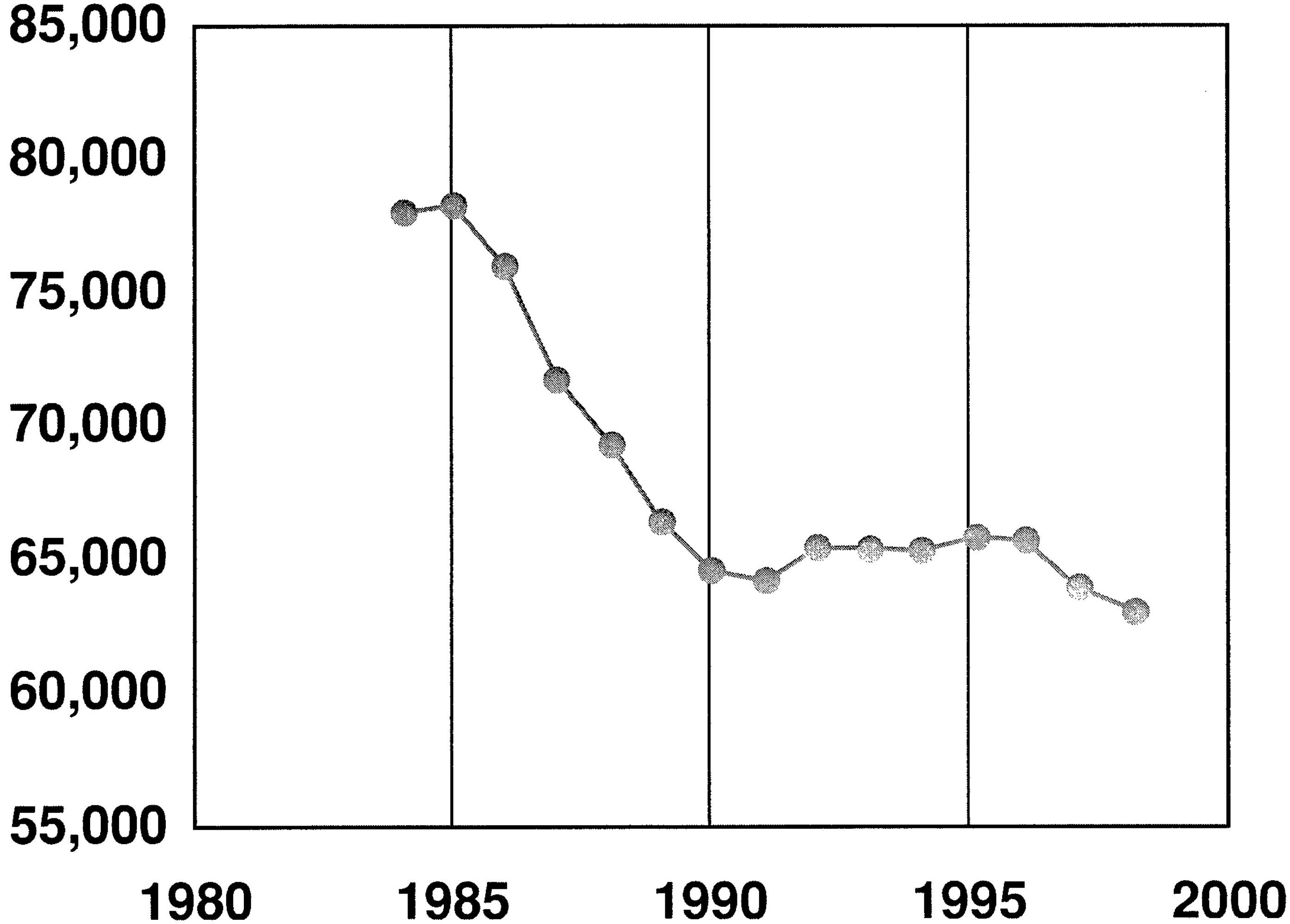
- Defense Capability
- Air Transport for Commerce
- Manufacturing Industry

INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

VSM-0382

Figure 4

Trends in Engineering Education in the U.S.



INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

VSM-0386

ISABE-2001-1002

COMPRESSOR AERO DESIGN AT GENERAL ELECTRIC BEFORE CFD

By
Leroy H. Smith, Jr.
Consultant to
GE Aircraft Engines
Cincinnati, Ohio 45215
U.S.A.

ABSTRACT

For the past 15 years, three-dimensional computational fluid dynamics (CFD) codes have been used extensively in the design process to custom tailor compressor blade shapes. Before these tools were available, designers employed one- and two-dimensional methods in a design system that was calibrated with experience. This paper describes the evolution at General Electric of these earlier methods.

The two main elements of the detailed aero design process are vector diagram establishment and airfoil design. Radial equilibrium solutions to obtain vector diagrams were employed at first only at stations between blade rows. By the late 1950s some transonic airfoil shapes were being custom tailored using internal blade station data from more complete radial equilibrium solutions. In the 1960s, rules and codes for shaping transonic passages were established, and by the 1970s, custom tailoring was done for subsonic blading as well.

The preliminary design layout of a new compressor involves selecting an annulus shape and blading overall proportions that will allow a successful detailed design to follow. This required establishment of stage loading limits that permit stall-free operation, and an efficiency potential prediction method for state-of-the-art blading.

As design methods evolved, the newer approaches were calibrated with data-match experience, a process that is expected to be always needed.

Key Words: compressor aero
compressor design

EXTENDED ABSTRACT

For the past 15 years, three-dimensional computational fluid dynamics (CFD) codes have been used extensively in the design process to custom tailor compressor blade shapes. Before these tools were available, designers employed one- and two-dimensional methods in a design system that was calibrated with experience. This paper describes the evolution at General Electric of these earlier methods.

The two main elements of the detailed aero design process are vector diagram establishment and airfoil design. Early designs employed large rotor inlet preswirl to keep tip Mach numbers subsonic, and with only slide rules and desk calculators at first available, clever means were devised to deduce the radially varying axial velocities needed to complete the vector diagrams. When high-speed digital computers became available in the 1950s, more elegant radial equilibrium solutions were employed, at first only at stations between blade rows but soon after at internal blade stations as well. For subsonic airfoils, NACA 65-Series thickness distributions on circular arc meanlines were used, and double-circular-arc profiles were used for transonic rotors. By the late 1950s some transonic airfoil shapes were being custom tailored using internal blade station data from the radial equilibrium solutions. In the 1960s, rules and codes for shaping transonic passages were established, and by the 1970s, custom tailoring was done for subsonic blading as well.

The preliminary design layout of a new compressor involves selecting an annulus shape and blading overall proportions that will allow a successful detailed design to follow. This requires establishment of stage loading limits that will permit stall-free operation, and an efficiency potential prediction method for state-of-the-art blading.

As the methods outlined above evolved, the new approaches were calibrated with data-match experience, a process that is expected to be always needed.

Propulsion Strategy for the 21st Century – A Vision into the Future

By

Dr. M.J. Benzakein
General Manager – Advanced Engineering
GE Aircraft Engines, Cincinnati, Ohio 45215
USA

Abstract

GE Aircraft Engines' (GEAE) commercial and military customers are striving toward products with low cost of ownership. This drives GEAE to high performance, light weight, low noise and low emission designs. GEAE's strategy is developing single-stage high-pressure turbine (HPT) machines for the narrow body/regional market, and two-stage HPT, high pressure ratio architecture for the wide body, long range operations. We have defined a two-step technology process with the TECH56 and the Ultra Efficient Engine Technology (UEET) programs. Work on fan, compressor, and turbine aerodynamics is in process.

Great strides to improve the environmental impact of aircraft engines are being taken. GEAE is working on NOx reduction with our TAPS combustor and on noise control with chevron nozzles.

The paper also describes GEAE's Global Engineering and University initiatives.

ISABE-2001-1006

SCRAMJETS AND SHOCK TUNNELS

By

R.J. Stalker and A. Paull
Department of Mechanical Engineering
University of Queensland
Brisbane, 4072, Queensland
Australia

Extended Abstract

Experience at the University of Queensland in using a shock tunnel to learn about scramjets is described. Studies of supersonic combustors in isolation were used to indicate that freestream dissociation did not affect the combustion energy release as much as expected, and scaling studies of a simple combustor established that the scaling law "pressure x length = constant" was valid for supersonic combustors. Raising the injection temperature of the hydrogen fuel increased the combustion length, while the change from central strut injection to sidewall port injection was found to make no observable difference to the combustion length. Finally, a shock tunnel – vitiated air facility comparison indicated that the two would yield the same results for a supersonic combustor.

Thrust/drag experiments with integrated scramjet models, using a specially developed stress wave force balance, indicated that combustion lengths shorter than those obtained in the combustor tests were possible but, nevertheless, the measured net thrust fell below expectations. This was ascribed to the effects of skin friction and to the pressure/size limitation of the shock tunnel.

An experimental study of skin friction in supersonic combustors indicated that it was unaffected by injection and combustion of hydrogen in the mainstream. However, experiments showed that injection and combustion of hydrogen in a turbulent boundary layer yielded very substantial skin friction reductions. Preliminary estimates indicate that combustion induced skin friction reduction has the capacity to considerably reduce the size of hypersonic scramjet vehicles.

ISABE-2001-1007

RESEARCH IN HYPERSONIC AIRBREATHING PROPULSION AT THE NASA LANGLEY RESEARCH CENTER

Ajay Kumar, J. Philip Drummond, Charles R. McClinton, and J. Larry Hunt
NASA Langley Research Center
Hampton, VA 23681-2199 (USA)

Abstract

The NASA Langley Research Center has been conducting research for over 4 decades to develop technology for an airbreathing-propelled vehicle. Several other organizations within the United States have also been involved in this endeavor. Even though significant progress has been made over this period, a hypersonic airbreathing vehicle has not yet been realized due to low technology maturity. One of the major reasons for the slow progress in technology development has been the low level and cyclic nature of funding. The paper provides a brief historical overview of research in hypersonic airbreathing technology and then discusses current efforts at NASA Langley to develop various analytical, computational, and experimental design tools and their application in the development of future hypersonic airbreathing vehicles. The main focus of this paper is on the hypersonic airbreathing propulsion technology.

Key Words: - Hypersonic Airbreathing Vehicles
- Scramjet Propulsion

ISABE-2001-1008

HIGH-FREQUENCY ROTATING INSTABILITIES IN AXIAL FLOW COMPRESSORS

By

Masahiro Inoue
Kyushu University
Department of Mechanical Engineering
Hakozaki 6-10-1, Higashi-ku, Fukuoka 812-8581
Japan

Extended Abstract

The high-frequency rotating instabilities of axial-flow compressors are phenomena that are encountered at low flow-rate operation prior to stall. They are usually caused by short wavelength disturbances rotating at a different speed of the rotor, and accompany with non-rotational components in pressure spectra. They do little harm to the overall performance of compressor, dissimilar to the usual rotating stall and surge, but they are responsible for the excitation of rotor blade vibration and noise generation.

This paper includes a review of compressor instabilities occurring at low flow-rate, and a presentation of high-frequency rotating instabilities. Through the review, the high-frequency rotating instabilities are distinguished from the common compressor instabilities: rotating stall and surge. Then, two kinds of the high-frequency rotating instabilities are presented on the basis of vortex consideration. The first one is caused by propagation of the multiple small-sized part-span stall cells. By means of a computer-aided measurement technique, it is found that the cell consists of a tornado-like vortex spanning from the blade suction surface to the casing wall ahead of the rotor, and rotates at about 70 % of the rotor speed. The second one is caused by unsteady behavior of the tip leakage vortex. The numerical simulations with analytical vortex-core identifications show that the unsteady behavior is closely related to the breakdown of tip-leakage vortex.

ISABE-2001-1009

Advanced Compressor Technology - Key Success Factor for Competitiveness in modern Aero Engines

Dr. Klaus Steffens
MTU Aero Engines GmbH
Dachauer Str. 665
D-80995 München

Abstract

Since day one of the gas turbine engine the compressor was the key component for the success and always required high development efforts and costs.

The importance of the compressors for modern high bypass ratio engines is demonstrated by the fact that 50-60% of the engine length and up to 40% of the manufacturing costs are covered by the compression system. The advances achieved allow engines to operate with core engine thermal efficiencies in the 50% area and propulsive efficiencies approaching 80%.

Integrally bladed rotors permit blade speeds significantly above conventional rotors and hence stage pressure ratios of >1,8.

A core compressor study with independent variation of the number of stages and aspect ratios from 0,6 to 2 gives insight into the relative weight, manufacturing and direct operating costs.

ISABE-2001-1010

Expanding the Horizons of Gas Turbines in Global Markets

Philip C. Ruffles
Rolls-Royce plc
PO Box 31, Moor Lane, Derby DE24 8BJ, United Kingdom

Abstract

Since its inception, the the ability to develop the gas turbine to meet customer's needs has depended on advances in key technologies and innovative thinking. The application and likely future development of gas turbines in the key sectors of aerospace, energy and marine areas are discussed, with particular consideration given to the use of aerospace derived technology to meet the needs of the industrial and marine sectors.

ISABE-2001-1011

A Comprehensive Approach to Engine Noise Reduction Technology

Dr. David E. Crow
Senior Vice President, Engineering
Pratt & Whitney
United Technologies Corporation
East Hartford, CT., USA

Abstract

In the late 1960s, Pratt & Whitney made a major leap in noise reduction when it introduced the first high bypass ratio turbofan, the JT9D. It has made steady, incremental progress every year since then. Today, airports and regulatory agencies are looking for additional major improvements in noise reduction, a trend that is likely to continue in the future. Pratt & Whitney is taking the lead in developing new technology to meet this need. However, the industry will need continued government funding, substantial investments in research and development, and consistency in rule making to continue meeting the expectations of its customers.

ISABE-2001-1012

TRENDS IN THE TECHNOLOGICAL DEVELOPMENT OF AEROENGINES : AN OVERVIEW

Christian Mari
Vice-President Research & Technology Snecma Moteurs
Rond Point G.Ravaud
77550 Moissy-Cramayel, France

Abstract

In the competitive market of aero engines, the engine manufacturer is faced to a highly demanding challenge: offer at an attractive price, products providing the highest benefit to the customers.
The challenge for people in charge of the research and technology strategy is that the technical needs may be strongly influenced by economical, political and environmental factors with a changing emphasis towards efficient, affordable or "green" engines.

To be efficient the strategy for technical progress must rely on the analysis of medium and long-term needs identified from the customers. These needs must then be translated into technical targets. The last stage is to define the research or demonstration programmes and to perform them successfully with the appropriate partners.

ISABE-2001-1013

"NASA Aeropropulsion Research: Looking Forward"

Jonathan A. Seidel
Arun K. Sehra
Renato O. Colantonio

National Aeronautics and Space Administration
Glenn Research Center
Cleveland, OH 44135 USA

Abstract

NASA has been working on developing new technology and system concepts to meet the requirements of aeropropulsion for 21st Century aircraft. The air transportation for the new millennium will require revolutionary solutions to meet public demand for improving safety, reliability, environmental compatibility, and affordability. This paper reviews the continuing turbine engine revolution and future propulsion system revolutions that will be required to achieve the dream of an affordable, emissionless, and silent aircraft. A number of new promising concepts, ranging from ultra-high bypass ratio intelligent engine to fuel cell-powered distributed propulsion system are also reviewed.

Key Words: Aeropropulsion, Intelligent Engine, Distributed Propulsion System

ISABE-2001-1014

EVOLUTIONS IN AIRCRAFT ENGINE DESIGN AND A VISION FOR THE FUTURE

Martin Jones,
Chief of New Projects Engineering, Rolls-Royce plc, Bristol, England

Phil Curnock,
Performance Team Leader, New Projects Engineering, Rolls-Royce plc, Bristol, England

Stephen John Bradbrook
Specialist Engineer, New Projects Engineering, Rolls-Royce plc, Bristol, England

Nigel Birch
Chief of Noise Engineering, Rolls-Royce plc, Derby, England

Keywords: *propulsion, aero-engine*

Abstract

This paper examines the evolution of gas-turbine propulsion system development over the last 50 years and the prospects for the future. Some 50 years of engine development will be reviewed and trends in particular attributes of engines will be discussed along with some likely limitations.

This period started with the use of the same un-reheated turbojet propulsion system for both military and civil applications. However, the differing emphasis on the requirements for combat aircraft and civil transport resulted in the evolution of different engine architectures, which supported their own technology requirements, as well as a set of common technologies.

The market drivers for civil engines have primarily been low fuel consumption and noise. This has led to un-reheated high bypass (> 5) ratio engines with high overall pressure ratio, Turbine Entry Temperature and component efficiencies.

The drivers for military combat engines have primarily been high specific thrust (thrust per unit airflow) and thrust-to-weight ratio. This enables high-speed flight and high maneuverability. The resulting engines generally have low bypass ratios (< 1), often with reheat, and with high Turbine Entry Temperature.

These changes have been supported by the development of a series of technologies: some common to both civil engine and military engine propulsion systems and others specific to the application.

In the future, civil market drivers are likely to also include reduced impact on the environment and continuing pressure to reduce the cost of ownership, namely, product unit cost, maintenance costs and reliability. The military combat scene is also likely to place greater emphasis on reduced cost of ownership but with the added requirement to control signatures, in particular radar cross section and infrared radiation emissions.

The paper concludes with a discussion of the technologies required to support these developing requirements.

ISABE-2002-1016

PRELIMINARY ENGINE DESIGN FOR AFFORDABLE CAPABILITY

M J Jones and S J Bradbrook,
New Projects Engineering, Rolls-Royce plc, Bristol, PO Box 3,

Abstract

This paper will examine the evolution of the military engine preliminary design process to support 'capability vs. cost' trades conducted at the weapon system level.

The engine is a major sub-system of all air vehicle assets making up a force mix. Changes in the engine capability, e.g. thrust/weight and specific fuel consumption, can be tracked through to air vehicle performance and ultimately to the force mix capability. In the same manner these changes will impact on the engine and air-vehicle life cycle costs and hence the total system costs.

Rolls-Royce has developed a preliminary design process to quickly assess changes in the engine: scale, thermodynamic cycle and technology level. The process quantifies the effect on engine performance, mass, size and cost. Cost includes: development, production and in-service support.

This process has been used to examine a range of engines both: 'all-new' and derivatives of existing engines. A key aspect of this work is the effect of technology on capability and cost. This has included the use of technology in the following areas: engine cycle, reduced weight, improved life and reduced cost. In addition, new applications, such as unmanned aircraft, have highlighted the need to consider new areas of technology e.g. long term storage.

The paper concludes with a look at an example of an engine trade study carried out and the scope for further studies.

ISABE-2001-1017

Recent Developments in U.S. Engine Noise Reduction Research

By

James Bridges, Edmane Envia, and Dennis Huff
NASA Glenn Research Center
Cleveland, OH
USA

Extended Abstract

Aircraft engine noise research in the United States has made considerable progress over the past ten years for both subsonic and supersonic flight applications. The Advanced Subsonic Technology (AST) Noise Reduction Program started in 1994 and will be completed in 2001 without major changes to program plans and funding levels. As a result, significant progress has been made toward the goal of reducing engine source noise by 6 EPNdB (Effective Perceived Noise level in decibels. This paper will summarize some of the significant accomplishments from the subsonic research performed over the past ten years. The review is by no means comprehensive and only represents a sample of major accomplishments.

While changing the engine cycle has always been a reliable way to decrease noise, the AST Program had goals to reduce noise from current turbofan engines. The development and validation of noise prediction tools were regarded as the key to providing low-noise design technologies. The intent was to develop fan and jet noise prediction methods and use them to identify noise reduction technologies that could be either retrofitted to an existing engine or applied to a new centerline engine design. The concepts were validated in model scale wind tunnel tests to confirm acoustic and aerodynamic performance. Many concepts have been tried and are being documented in NASA reports and society papers as the AST Program concludes this year. Some of the concepts are being tested in static engine and flight tests.

Work will continue under the new Quiet Aircraft Technology (QAT) Program that started in 2001 and will end in 2005. This program will provide technologies for meeting one of NASA's strategic objective goals to reduce aircraft noise by 10 dB relative to 1997 subsonic aircraft levels.

ISABE-2001-1018

ACTIVE COMBUSTION CONTROL: RECENT R&D AND OUTLOOK FOR IMPLEMENTATION

Klaus C. Schadow
Consultant with Strategic Analysis, Inc.
San Clemente, CA 92673
E-mail: schadowkc@home.com

and

Ken H. Yu
Aerospace Engineering Department
University of Maryland
College Park, MD 20742
E-mail: yu@eng.umd.edu

Significant progress has been achieved over the last 5 years in implementing active combustion control (ACC), ranging from sub-scale experiments with different types of simulated combustors to full-scale tests with aeroengines and field demonstration with industrial gas turbines. This paper reviews recent R&D activities, and provides an outlook for ACC implementation in various systems. In order for ACC to become an enabling technology, the future combustor requirements and demand must be considered. Furthermore, improved understanding of combustion dynamics under practical conditions, developing high-authority actuators for controlling the combustion processes, and demonstrating robustness of control models at full-scale are needed.

Keywords: active control, combustion dynamics

Experimental and Theoretical Investigation on 2D and 3D Parallel Hydrogen/Air Mixing in a Supersonic Flow

P. Gerlinger, P. Kasal[*], P. Stoll[*] and D. Brüggemann[+]

Institut für Verbrennungstechnik
DLR-VT, 70569 Stuttgart, Germany
***Institut für Thermodynamik der Luft- und Raumfahrt**
Universität Stuttgart, 70550 Stuttgart, Germany
[+]Lehrstuhl für Technische Thermodynamik und Transportprozesse
Universität Bayreuth, 95440 Bayreuth, Germany

Abstract

An experimental and numerical investigation of the nonreactive hydrogen/air mixing processes in cold Mach 2 flows will be presented. Planar and lobed strut injectors are compared by use of performance parameters. In case of the lobed injector mixing is enhanced by the generation of strong streamwise vorticity. The experimental investigations include schlieren photographs, wall pressure measurements and Raman measuremens of major species concentrations. With these results an improved strut design is proposed that is based on three dimensional numerical simulations.

Keywords: supersonic combustion, mixing, lobed injector

ISABE-2001-1020

Ramp Nozzle for Supersonic Mixing and Combustion

Hari,S. and Kurian, J.

Department of Aerospace Engineering

Indian Institute of Technology, Madras. India

A supersonic ramp nozzle that features five swept ramps, is studied for its freejet characteristics, jet structure and confined momentum and thermal mixing performance. The study was conducted experimentally at a nozzle Mach number of 1.7.The conventional measurement techniques for pressure and temperature were employed. Jet structure was observed using schlieren . Laser Light Sheet (LLS) method was used to obtain cross sectional views of the jet due to the introduction of ramps. Two factors for quantifying mixing performance and the pressure drop are defined. The ramps are found to augment the mixing behavior of conventional supersonic nozzles though with considerable pressure losses. The acoustic measurements revealed that the ramps resulted in increasing the screech component of the supersonic jet.

Key words: mixing ,ramp nozzle

Contact Author: Dr. Job Kurian
Professor
Department of Aerospace engineering
IIT Madras.
E mail:Kurian@aero.iitm.ernet.in

ISABE-2001-1021

Effects of Trailing-Edge Geometry of Splitter Plates on the Growth Rate of a Compressible Mixing Layer

Mikiya ARAKI, Yuichi OGURA, Mitsuhiro TSUE and Michikata KONO
Department of Aeronautics and Astronautics, University of Tokyo,
7-3-1 Hongo, Bunkyo-ku, Tokyo 113-8656, JAPAN

Co-Authors : Jun OSAKA, Shinji NAKAYA, Mitsutoshi OSHITA
(University of Tokyo, JAPAN)

Abstract

Effects of the trailing-edge geometry on the growth rate of a compressible mixing layer were investigated experimentally. Sine-curved trailing edges were used for splitter plates. Subsonic air was in parallel injected into Mach 1.78 air stream. Two parameters were effective on mixing enhancement, namely the phase difference between upper and lower trailing-edge sine curves and the maximum angle of trailing-edge sine curve, θ_{trail}. Using out-of-phase trailing edges, secondary vortex structures are considered to be formed, resulting in remarkable mixing enhancement. Around the θ_{trail} of 65 degrees, the maximum growth rate was achieved, increased by about 80 %.

Key words : Compressible flow, Mixing enhancement, Secondary instability, Scramjet

SUPERSONIC MIXING AND COMBUSTION IN CHARACTERISTICS IN SCRAMJET ENGINES

S.N. Tiwari*, A.A. Taha†, T.M. Abdel-Salam†, and T.O. Mohieldin‡
**Eminent Professor/Scholar, Dept. of Mechanical Engineering, Old Dominion University.*
†*Graduate Research Assistant, Dept. of Mechanical Engineering, Old Dominion University.*
‡*Associate Professor, Dept. of Engineering Technology, Old Dominion University.*

Extended

A numerical study is conducted to study the mixing and combustion processes in scramjet engines. The study is divided into two parts. In the first part, a compression wall-mounted ramp has been selected as a fuel injector to investigate the effect of the ramp side angle in the mixing process of scramjet engine using unstructured grids. A three dimensional configuration has been used with ramp-side angles of 0, 5 and 10 degrees. Numerical results for this part are obtained with a grid size approximately 300,000 nodes.

In the second part, mixing and combustion characteristics of ethylene in scramjet engines are investigated. The configuration studied features the existence of a generic rearward-facing step in the upper longitudinal wall. The effect of pilot injection on the combustion of ethylene fuel injected normal to the incoming supersonic air stream is investigated. The results for ethylene supersonic combustion show the typical features of such flowfields. The existence of a wedge downstream of the rearward-facing step shows a good potential towards enhancing the fuel-air mixing in addition to helping in initiating and stabilizing the main flame. This part of the study is still underway to investigate the combustion flowfield of ethylene, propane and kerosene injection with pilot hydrogen. The motivation is to provide a comprehensive study to address the possibility of using hydrocarbon fuels in the mid-speed range of the hypersonic flight. Figure 1 shows the schematic diagram for the second case study.

Figures 2-5 present part of the results for the first case study. The injectant mole fraction cross flow distribution for the three side sweep angles is shown in Fig.2. Figure 3 shows the streamwise vorticity illustrated by the velocity vectors in two different cross flow planes. The mixing rate of the three ramps is illustrated in Fig.4. The figure shows the axial decay of the maximum injectant mole fraction for the three side sweep angles. Figure 5 shows the axial decay of the injectant concentration.

Selected results of the second case study are presented in Figs. 6-9. Figure 6 shows the static pressure contours at the symmetry plane while the temperature contours at the same plane are presented in Fig 7. The flame areas at the sides of the normal injection are clearly seen in the CO_2 mass fraction contours presented in Fig. 8. Figure 9 shows the distribution of the average static pressure and temperature at the upper wall along the combustor length. The static pressure distribution belongs to the left y-axis while the static temperature distribution belongs to the right y-axis.

ISABE-2001-1023

MIXING OF COAXIAL TRANSVERSE JETS OF HYDROGEN AND AIR INJECTED INTO SUPERSONIC FLOW

Takahiro Bamba and Shojiro Kaji

University of Tokyo

Department of Aeronautics and Astronautics

Tokyo 113-8656 Japan

Abstract

Although a number of studies conclude that swirled injection is effective method in supersonic mixing, the degree of mixing achieved is still insufficient for application. Searching for a way to enhance mixing utilizing swirl, coaxial transverse injection of hydrogen fuel and air into supersonic air stream is numerically investigated. The results show that air injected from inside holds up the entire jet to penetrate deeply into and spread through the main stream, which leads to significant improvement in mixing efficiency. Effect of swirl and pressure ratio of two injected gases in mixing is also examined.

Key words: mixing enhancement, coaxial jets, swirl, scramjet engine

ISABE-2001-1024

INVESTIGATION OF BASE PRESSURE BEHIND THE INJECTOR SECTION IN A SUPERSONIC COMBUSTION CHAMBER

Paul J. Ortwerth, Atul B. Mathur
Boeing – Rocketdyne Propulsion and Power, USA
Marat A. Goldfeld, Roman V. Nestoulia
Institute of Theoretical and Applied Mechanics, Russia

Additional Contributor: Alexey V. Starov

Abstract

The results of experimental research of base pressure at flight enthalpy Mach numbers of 6-9 in a direct connect supersonic combustion chamber with multi-injector section are presented. Four ramp fuel injectors with sidewall ratio b/h=1 had supersonic fuel nozzles and a separate system of fuel injection into the base region. Base injection was investigated in two configurations, a distributed axial array in the base (#1, #2) and rim injection (#3) into the base region was performed through 112 sonic nozzles. The tests were carried out at Mach numbers 3 and 4 with and without a tunnel supply pressure multiplicator. Room temperature hydrogen fuel was used. The effect of various ratios of fuel injection rates through the main and base fuel supply system on the fuel ignition and combustion was studied. A combustor phenomenon called "kindling" was observed in all tests, which separated the combustor dynamics into an initial period and a later intense combustion period This had an equally intense effect on the base pressure. The base pressure measured just in the injector section is presented as a function of the fuel mass rate injected into the base region. The base pressure measured in the injector section rises to more than twice the pressure before the injector section with intense combustion of hydrogen. Results obtained showed that variation of the amount of fuel injected into the base region has no substantial influence on the combustion intensity and other characteristics of the combustion. In these investigations it was found that base pressure is primarily controlled by overall equivalence ratio.

Key words: supersonic combustion, fuel injection, base pressure, drag/thrust

Contact Author Sheet, Paper 2001-24
"Investigation of Base Pressure Behind the Injector Section in a Supersonic Combustion Chamber"

Dr. Atul B. Mathur
Boeing – Rocketdyne Propulsion & Power

atul.b.mathur@boeing.com

(818)586-1626
(818)586-0579 - Fax

Biography of Presenting Author, Dr. Atul B. Mathur
Paper 2001-24
"Investigation of Base Pressure Behind the Injector Section in a Supersonic Combustion Chamber"

Dr. Atul Mathur has over 20 years experience in high speed propulsion and aerothermodynamics of propulsion systems. After completing undergraduate engineering studies at the Indian Institute of Technology in New Delhi, India, he pursued graduate studies in Aerospace Engineering at Virginia Polytechnic Institute and State University in Blacksburg, Virginia. He obtained his Ph.D. in Aerospace Engineering in 1983. He worked as a Principal Research Investigator and Adjunct Professor in the Department of Aeronautics at the Naval Postgraduate School in Monterey, California for five years before joining the Boeing company in the Rocketdyne Propulsion and Power division as a senior engineer in 1987. He was closely involved with the National Aerospace Plane program till 1992, first as a Principal Engineering Specialist, and then as the Manager of the Airbreathing Combustion and System Performance group. During this time, he obtained a Master's degree in Technology Management from Pepperdine University in Malibu, California. He is currently Principal Manager, Aerothermodynamics, in the Advanced Analysis group at Rocketdyne.

ISABE-2001-1025

Gamma Titanium Aluminide for Static Structures

by

Srini R. Gowda,GE Aircraft Engines,
Joseph E. Grady, NASA Glenn Research Center
and
Robert Warburton, Pratt & Whitney Aircraft Engines, USA

Abstract

Future High Speed Civil Transport (HSCT) air-vehicles will demand very high fuel efficiency and very low noise levels for it to be commercially viable. High fuel efficiency requirements come from the economics of being able to compete in the commercial aviation market in par with current operating costs and the low noise requirements are necessitated by the anticipated noise requirements of future which are likely to be more stringent than the current regulations. Additionally, low NOx is also a critical requirement.

Under a NASA-USA sponsored " High Speed Civil Transport (HSCT) " Program, the engine and nozzle system design resulted in the total weight and size of the nozzle component to be greater than the propulsion core. What was required for the nozzle wall was a material along the densities similar to Aluminum alloys but with stiffness and temperature capabilities comparable to that of Titanium. Many high temperature materials were evaluated for this specific application and Gamma TiAl was chosen due to its low density, high temperature and high stiffness capabilities. The use of this relatively new material for the nozzle application required development of manufacturing processes like structural castings, forming and joining. This paper describes some of the unique processing methods developed under the HSCT Program for Gamma TiAl in its use as a static structural material.

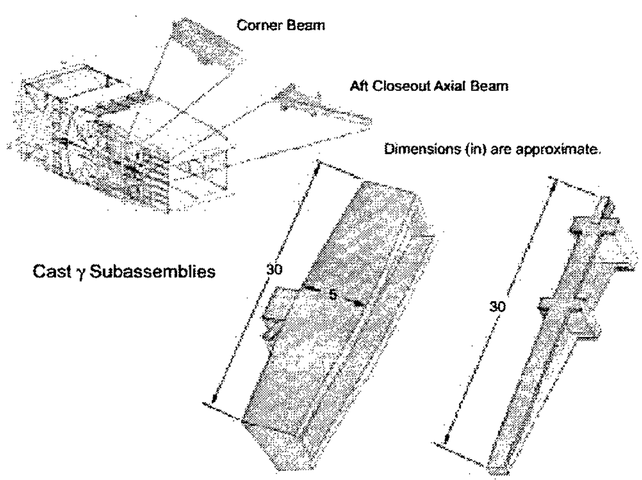


Figure 1. Generic Design Nozzle Sidewall Showing Location of Closeout Beam and Corner Beam Subsegments

Key Words : **Gamma Titanium Aluminide, Processing of Gamma TiAl.**

ISABE-2001-1026

GAMMA TITANIUM-ALUMINIDE TECHNOLOGY WITHIN THE ADVANCED PROPULSION COMMUNITY

Dennis M. Dimiduk
Air Force Research Laboratory, Materials and Manufacturing Directorate
AFRL/MLLM Bldg. 655, 2230 Tenth Street
Wright-Patterson AFB, OH 45433-7817, USA

Abstract

This manuscript summarizes the operational paradigms for the propulsion design and materials communities over the last 15 years. These are important since it is not apparent that the changes within each community are understood by the other; yet the two communities are interdependent for creating advanced propulsion products. Based in part on this background, the progress of gamma titanium aluminides is analyzed, problems are highlighted, and the status synopsized. Finally, some suggestions are made for the future.

Future growth of turbine engines and transportation systems depends upon the continued development of high-temperature, lightweight materials. Further, substantial fuel savings in commercial aircraft and power generation can be achieved by using new materials which can provide this temperature increase. It has been suggested that intermetallic alloys could be part of the solution for these required efficiency gains. However, two things must be remembered in making such a suggestion: i) the current materials of construction exhibit a balanced set of properties and strength is only one of these; and ii) intermetallic alloys are not simple extensions of conventional alloys. The introduction of a new material for use in structural components can be a complex process involving both the materials and processes community and the structures design community. In the present business climate there must be a quantifiable benefit from the outset. Often this was associated with a performance improvement based on increased properties, but cost reduction and reliability increases are now the important drivers. Nonetheless, the last decade has brought dramatic growth in fundamental understanding of the properties of gamma alloys, and growth in the technological aspects of producing them. Laboratory-scale demonstration of an attractive balance of alloy properties opens revolutionary opportunities for weight reduction, and perhaps increased operating temperatures in aerospace systems, but at a price. That price may be the need for refined design approaches, and an involved, time-consuming, and expensive sequence to calibrate the design systems and to build confidence in the material. The insertion of these and other new materials into service will require co-operative efforts with design and materials engineers working together in a systems fashion—moving beyond the historical paradigms of these communities.

Key Words: High Temperature Materials, Titanium Aluminides, Alloy Development

ISABE-2001-1029

MANUFACTURING PROCESS OF HOOP REINFORCED TITANIUM MATRIX COMPOSITE RING

A.Kono, T.Yamada, A.Fukusima and M.Hirota

Nagoya Aerospace Systems
Mitsubishi Heavy Industries, Ltd.
10, Oye-cho, Minato-ku, Nagoya, 455-8515, JPN

Continuous fiber reinforced titanium matrix composites are attractive as potential aerospace structures due to their high specific mechanical properties and improved high temperature capability. Particularly, when the fan rotor ring is reinforced with TMCs, weight saving of more than 30% has been predicted, because this ring is able to carry the high hoop stress of TMC longitudinal properties. In the government support program, manufacturing process of TMC ring is being researched and developed.

As for fabrication method for TMC ring or disk components, some of fabrication methods have been proposed. In this paper, fiber winding and spraying process was adopted because this is cost effective for thin TMC ring parts of fan rotor

The major problem of manufacturing TMC ring is fibers fracture during consolidation. During hot isostatic pressing (HIPing) of ring components, it is inevitable that the radius expands, which can result in considerable tension in the fibers, leading to fibers fracture. Nonetheless there are a lot of examples of producing the TMC ring components, but few papers has been reported the mechanism about the fiber damage during heating to the HIPing temperature and HIPing condition.

In this paper, fibers fracture is investigated by the analytical and experimental approach. During heating procedure in the HIPing process, fibers fracture is affected by the thermal expansion of the core ring material that is set inside the TMC preform. To prevent this fracture, the material of low thermal expansion coefficient like CP-Ti and Ti-6-4 is adopted for core ring, and heating condition in HIPing process is improved to make temperature distribution uniform. Appearance of the TMC ring produced by this improved process, at the center of the ring few of the fiber fractures observed. It is clear that the thermal expansion of the core ring material affected fibers fracture.

ISABE-2001-1030

The Inadequacy of Safe-Life Prediction: Aero-Engine Fan and Compressor Disk Cracking

B.J. Wicks and R.A. Antoniou
Aeronautical and Maritime Research Laboratory
Defence Science and Technology Organisation
Department of Defence
Box 4331 GPO
Melbourne 3001
Australia

Abstract

The use of a safe-life methodology to ascribe a replacement interval to gas turbine engine components has been used extensively for over 40 years. However there are inherent flaws within the methodology, resulting in significant under-utilisation of component lives, and an inability to account for rogue flaws and other non-representative factors. This paper will present three examples where the safe-life approach was inadequate in predicting the safe working life of critical engine components. These examples illustrate the complexity of the processes that have to be taken into account to produce realistic life estimates.

ISABE-2001-1031

ON-LINE VALIDATION OF MEASUREMENTS ON JET ENGINES USING AUTOMATIC LEARNING METHODS

Pierre Dewallef **Olivier Léonard**

University of Liège
LTAS (Aerospace Laboratory)
Turbomachinery Group
Chemin des chevreuils 1 (B52/3)
4000 Liège - BELGIUM

p.dewallef@ulg.ac.be **o.leonard@ulg.ac.be**

Abstract

Nowadays, turbine engine tests are processed using an open loop, i.e. the measurements are verified and treated *a posteriori*, sometimes weeks or months after the end of the test. The scope of the present project is to develop a new methodology which enables real time detection of faulty measurements and the suppression of the source of these faults during the test.

The validation of the measurements is achieved by a "robust" parameter identification. Such a method is called robust in the sense that it can cope with a lot of faults (20 to 30% of the number of measurements). The robustness is brought by a distribution of the measurement noise, introduced by Huber, that takes into account the possibility of faults.

Since the scope of a parameter identification is to find the set of parameters which has most likely generated the measurements observed on the process, this leads to an optimisation problem that has to be solved for the parameters. The measurements are linked to the parameters through a non-linear model, leading to a large system of equations for modern jet engines. If no model of the process can be made available or if the model is too complex to allow real time validation, automatic learning methods may provide a solution: either a model is generated, directly based on the measurements (on-line learning), or a database is generated, based on the existing (but expensive) model, the database being used to build a statistical model (off-line learning).

Neural networks seem to be very suitable for modeling the behavior of turbojets, avoiding the resolution of a time-consuming non-linear system. In this paper neural networks are tested to generate a model based on the measurements generated by the model of a single flow, single spool and variable geometry nozzle turbojet. Only the off-line learning approach will be considered.

Keywords: robust identification, neural networks, measurement validation.

SETTING UP A BELIEF NETWORK FOR TURBOFAN DIAGNOSIS WITH THE AID OF AN ENGINE PERFORMANCE MODEL

By

C. Romessis, A. Stamatis, K. Mathioudakis
Laboratory of Thermal Turbomachines
National Technical University of Athens
PO BOX 64069, Athens 15710, Greece

Abstract

This paper presents a method for building Bayesian Belief Networks (BBN) for gas turbine performance fault diagnosis. Building a BBN requires information of stochastic nature. It is a common practice to extract this kind of information from statistical analysis of large data sets. In the field of gas turbine diagnostics, though, such data are usually hard to find. With the present method, the required information is extracted from an engine performance model. In this way, stochastic information, expressing the probability of whether a series of events occurred or not, can be extracted by a deterministic model and does not depend on, hard to find, flight data of different faulty operations of the engine.

The diagnostic problem is first described. Some basic concepts of BBN, though briefly described, are also presented in relation to turbofan engines.

A detailed description of the proposed way to set-up a diagnostic BBN from an engine performance model follows. Several simulated, but realistic, fault cases are then used for inference with the constructed network.

Inference with BBN showed that such a network is very reliable, since in the 96% of the cases where a fault was detected, it was detected correctly. Only a 4% of the cases was attributed to a wrong fault. In some cases, the network was not 'sensitive' in the presence of a fault, since it did not detect any fault at all. Further, preliminary work, though, shows that the 'sensitivity' of the network can be increased.

It is shown that building a BBN, based on information provided by an engine performance model, is feasible and can be efficient as well.

Keywords: Gas turbine Fault Diagnostics, Jet Engine Monitoring, Bayesian Belief Networks, Probabilistic Expert Systems

ISABE-2001-1033

Life consumption monitoring of the Walter M601E turboprop engine

By

Jaromír Lamka and František Sedlář
Aeronautical Research and Test Institute (VZLU, a.s.),
Beranovych 130, 199 05 PRAGUE 9 – Letnany
Czech Republic

Extended Abstract

The following paper describes briefly a monitoring system for service life consumption of the Walter M601E turboprop engine having a take off power of 595 kW installed on a two-engine L 410 UVP-E aircraft. Practical experience gained during its operation is discussed.

The monitoring system includes an algorithm to evaluate flight data from the viewpoint of cycle life usage of rotational parts, exceeding of limited operational loads, trends towards changes in measured parameters, evaluation and specification of operational periods, and service information. In the algorithm the safe life concept is applied. The algorithm consists of methodology of estimation as developed and used at ARTI during the design, certification and operation of the M601 engine. The computer program algorithm for low cycle fatigue life usage calculation of engine primary parts after one flight cycle was based on experience and an appropriate theoretical preparation. To analyze a variable-amplitude loading the rainflow analysis was used, which gives the most correct results according to previous experience. The flight data for computer program evaluation were obtained through an L 410 UVP-E aircraft that has been operated on both regular and irregular flights. The 19-seat aircraft is designed for economical flights and is able to operate on unpaved airfields. Aircraft onboard equipment has been ensembled by a new digital recording system for measuring and storage of necessary engine parameters and their following processing.

The flight data measured are processed at a ground evaluation station by the ENG 410 computer program. The program processes the flight data files automatically one after another - usually it is so that one flight data file describes one flight cycle. First the program will carry out data distribution to allocate data in a data base of basic statistic information, e.g. total time of engine run, total time of flights, number of takes-off, number of interrupted landings, etc. Histograms of the exceeded engine parameters are stored in the data base, too. Second, the ENG 410 program will calculate the life usage of the primary parts. The part of the ENG 410 program which calculate the life usage with respect to low cycle fatigue was verified to give very good results. Now the verification of the ENG 410 second part for calculation of hot engine parts life usage with respect to creep is carried out. Life usage under the damage tolerance concept is under development.

The results obtained by the designed monitoring system allow us to better understand the impact of different ways in which the engine is actually used during service. The simple civil operation of the L 410 UVP-E aircraft with M601 engines varies a great deal and consequently the life usage of the individual primary engine parts is very different from the classic engine life usage following i.e. a fixed interval maintenance scheme. Nevertheless we can unambiguously say that the monitoring systems previously applied to military airplanes, then big civil aircraft, are finding their applications to the low power category engines too i.e. up to 1 MW.

It will be interesting to compare our results with experience of other commuter operators.

ISABE-2001-1034

TURBOSHAFT ENGINE CONDITION MONITORING BY BAYESIAN IDENTIFICATION

L. Biagioni[†], **and R. Cinotti**[**]
Centrospazio-CPR, I-56014 Pisa, Italy

L. d'Agostino[*]
Dept. of Aerospace Engineering, University of Pisa, I-56100 Pisa, Italy

Abstract

The application of a *Maximum Likelihood Estimate* technique has been implemented in an Engine Condition Monitoring framework for a small turboshaft for power generation purposes. The turboshaft has been modeled in a fully non-linear way, by using actual turbomachine performance maps obtained from the manufacturer: the accuracy of the simulation proved to be very good with respect to real operating data. The model was used both to generate sample synthetic dataset (by adding Gaussian noise to the selected outputs-measurements) and as the core computational engine in the identification process. The results obtain show the very good robustness of the proposed identification process, and its capability of dealing with noisy or even malfunctioning transducers. This capability is provided by the possibility of determining a mathematically sound statistical framework, which is not only capable of identifying the most likely fault configuration but also to indicate the confidence level with which the identification is performed.

ISABE-2001-1036

OPTIMAL MEASUREMENT PARAMETER SELECTION OF TURBOPROP ENGINE FOR BASIC TRAINER USING GPA APPROACH

By

Changduk Kong, Jayoung Ki
Chosun University
Department of Aerospace Engineering
#375 Seosuk-dong, Dong-gu, Kwangju, Republic of Korea
Phone : 82-62-230-7188, E-mail : cdgong@mail.chosun.ac.kr

Extended Abstract

A program for steady state performance simulation and diagnostics of the turboprop engine (PT6A-62), which is the power plant of the first indigenous military basic trainer KT-1 in Republic of Korea, was developed.

The steady state performance analysis program was evaluated by both the performance data given by the engine manufacturer and the calculated results by GASTURB program. The performance analysis was performed at various flight velocities altitudes in the uninstalled and installed conditions. In the investigation, the SHP decreased as the altitude and flight speed increased, but the SFC increased at the same conditions. Furthermore, it was confirmed that the developed program has acceptable accuracy because of the error within 3% between GASTURB and the developed program.

The linear and nonlinear GPA techniques for diagnostics of turboprop engine were studied. An analysis for performance degradation was carried out by the changes of the performance degradation rate of independent parameters and the number and kinds of dependent parameters. The analysis results by the linear GPA were compared with those by the non-linear GPA. Moreover an analysis by the nonlinear GPA was performed with an iterative method, which the performance degradation rates of independent parameters were divided into same intervals, and theses results were compared with those by the Escher method.

In this investigation, the following conclusions obtained. The nonlinear GPA, which the same interval performance degradation rates of independent parameters, was not effective relatively to the linear GPA. However in case of the large performance degradation rate, the Non-Linear GPA method is much better than the linear GPA. As the number of the measurement parameters increased the RMS error decreased generally. Therefore if the measurement parameters may be properly selected a economic and reliable diagnostics can be realized.

ISABE-2001-1038

RECENT OUTCOMES IN COMPUTATIONAL DESIGN OF SUPERSONIC AIR INTAKES

ARNAUD GAIDDON*
EADS AEROSPATIALE MATRA MISSILES
Computational Design Department
2, rue Béranger BP 84
F-92320 CHATILLON Cedex, France
Tel: (33) 1 47 46 24 35
Fax: (33) 1 47 46 40 51

ABSTRACT

The design of supersonic missile inlets mostly relied on heavy and costly wind-tunnel tests. The recent progress in the fields of CFD and automated optimization allow us to propose a global design methodology that conducts to actual improvements in terms of productivity.

This paper gives a survey of the recent outcomes in the field of supersonic inlet simulation and optimization. Recent methods such as Navier-Stokes computations or genetic algorithms are now reliable enough to be intensively used in an industrial context. The practical design methodology is then explained, illustrating how both numerical and experimental approaches are linked to obtain the best synergy. The application to the French VESTA missile proves the improvements in terms of quality and reduction of costs.

ISABE-2001-1039

HYPERSONIC AIR INTAKE PERFORMANCE IMPROVEMENT THROUGH DIFFERENT BLEED SYSTEMS

S. Pandian, Joyas Jose, M.M. Patil and P. Srinivasa
Vikram Sarabhai Space Centre, Trivandrum, India.

Abstract

Air intake plays crucial role in the performance of air breathing vehicles. At high Mach numbers air intake performance is greatly determined by the type of boundary layer bleed system, location of bleed and extent of bleed. It is a challenging task to optimise the bleed system. A rectangular cross section air intake was designed based on in-viscid calculations. This air intake subscale models were tested in Hypersonic Wind Tunnel of Vikram Sarabhai Space Centre, Trivandrum at Mach number 5. The air intake performance was characterised through pressure measurement and flow visualisation studies. The performance improvement was attempted through different bleed systems. It is found that the base line configuration pressure recovery is 1/3 of the ideal recovery due to viscous effect. The corner boundary layer bleed improves the pressure recovery by about 35 % of the base line configuration at an optimal gap of about 1.0mm. Whereas the improvement in mass flow is marginal. The ramp boundary layer bleed improves the pressure recovery, but to a lesser extent. The corner bleed is observed to be better than the ramp bleed configuration from over all performance point of view.

Key Words : Hypersonic, Intake, Bleed

ISABE-2001-1040

Experimental studies on isolated supersonic air-intake models of a typical air-breathing launch vehicle

G. K. Suryanarayana*, A.E. Sivaramakrishnan#, Pankaj Priyadarshi# and V. R. Ganesan#
*National Aerospace Laboratories, Bangalore, India
#Vikram Sarabhai Space Center, Trivandrum, India

Abstract

The performance of two supersonic air-intake configurations S1 and S2 has been extensively analyzed through qualitative and quantitative measurements at Mach numbers in the range 1.8 to 3.0. The exit area of the intake was varied during the tests using a butterfly valve, which was controlled using a PC. The performance of S2 was found to be comparable with that of a standard supersonic intake, with characteristic features of supercritical and subcritical behavior whereas S1 configuration did not indicate any critical condition. The critical condition for S2 was found to occur when the exit area was about 1.24 times the throat area at M = 3.0. The measured total pressure recovery with S2 was found to be marginally higher than that for S1, whereas the mass flow rate through the intake showed considerable improvement (e.g., 11% at M = 3.0 and 19% at M = 2.0 at maximum pressure recovery condition). Similar improvements were found with S2 configuration at other Mach numbers also. It is proposed that an Intake Performance Index (product of the pressure recovery and mass flow efficiencies), as a function of the back-pressure may be used to compare the efficiencies of different intake configurations. In terms of this index, S2 configuration was found to be than the S1 configuration at all the Mach numbers tested.

ISABE-2001-1041

INTAKE AERODYNAMICS INVESTIGATIONS BY CFD AND CONNECTED INLET TESTING

Ulf Johansson and Jonas Larsson
Aerothermodynamics department
Volvo Aero Corporation
Trollhättan, Sweden

Abstract

A connected type of aircraft intake test rig has been fabricated and tested. The rig is used as a base for technology programs in the intake aerodynamics field. A combination of CFD (Computational Fluid Dynamics) and testing is used in the program The purpose with the program is to gain experience in intake aerodynamics and in the future to develop test and CFD methods for intake aerodynamics analysis and problem solving.

ISABE-2001-1043

PREDICTION OF NO_x EMISSIONS IN GAS TURBINE COMBUSTORS INCLUSIVE OF THE N_2O CONTRIBUTION

D. Lentini *
Dipartimento di Meccanica e Aeronautica
Università degli Studi di Roma "La Sapienza"
Via Eudossiana 18, I–00184 Roma, Italy

Abstract

A model is presented to account for NO emissions due to both the thermal and the nitrous oxide pathways, and to quantify emissions of N_2O itself. The model is applied to a hypothetical combustor operating at typical take–off conditions, as high–pressure conditions are expected to emphasize the role of the nitrous oxide pathway. Predictions suggest that this pathway may turn out to be a significant source in such conditions.

ISABE-2001-1044

NOx Emissions and Autoignition in a Lean Premixed Prevaporized Tubular Combustor at Inlet Air Temperatures up to 1050 K

Shigeru Hayashi and Hideshi Yamada

National Aerospace Laboratory
Aeroengine research Center
Chofu, Tokyo 182-8522 JAPAN

Abstract

The NOx emissions, flame stability and autoignition in a premixer for an SST turbojet combustor were investigated at atmospheric pressure. An air-blast fuel nozzle was used to premixed kerosene with the swirling combustion air in the premixer and the air velocity was varied in a range to modify the degree of atomization. A better trade-off between low NOx emissions and high combustion efficiency was achieved when air velocity was higher. Flame stabilization by autoignition was seen in the combustion chamber at temperatures higher than about 1020 K though no autoignition in the premixer was experienced even at the most severe conditions.

LPP combustion, premixer, NOx emissions, autoignition

ISABE-2001-1045

KINETIC COMBUSTION NEURAL MODELLING INTEGRATED INTO CFD

G. Cerri, V. Michelassi, S. Monacchia, P. Boatto
Dipartimento di Ingegneria Meccanica e Industriale
Università degli Studi Roma Tre
Roma, Italy
(e-mail: michelas@uniroma3.it)

Abstract

This paper presents a successful attempt to replace a traditional chemical kinetics model with a new model based on Neural Networks. The chemical kinetics model allows a detailed description of the combustion of methane in air, which is preliminary, analyzed in details and simplified. The chemical kinetics model is used to prepare a large data set to train a Neural Network, which reproduces the chemical kinetics. The results of the Neural model compare favorably with those of the kinetics model. Both the traditional chemical kinetics model and the neural model are cast into a commercial CFD tool. The results prove that the two combustion model approaches are of comparable accuracy, but the Neural Model is approximately 40 times faster.

ISABE-2001-1046

Evaluation of Nitric Oxide Kinetics for Gas Turbine Combustion

R. V. Ravikrishna
Assistant Professor, Dept. of Mechanical Engineering
Indian Institute of Science
Bangalore – 560012, India

Normand M. Laurendeau
Professor, School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907, USA

Abstract

Nitric Oxide kinetics in laminar, counterflow diffusion flames at pressures from 1 to 5 atm are assessed by comparing predictions of NO concentration ([NO]) using comprehensive chemical kinetic mechanisms with highly quantitative laser-induced fluorescence (LIF) measurements. The most recent Gas Research Institute (GRI) mechanism, version 3.0, is used and is compared with the older version 2.11 mechanism. Results indicate that there is significant improvement in the predictive capability of the GRI 3.0 mechanism over the 2.11 mechanism, with regard to predicting the magnitude of [NO] as well as the trend of [NO] variation with pressure. However, there is still substantial difference between [NO] predictions and measurements, especially at lower pressures, indicating a need for further refinement of NO kinetics.

Key Words: Nitric Oxide Kinetics, Laser-induced Fluorescence (LIF), Counterflow Diffusion Flames

ISABE-2001-1047

Experimental and Numerical Simulations of Photochemical Reactions in the Ozone Layer with NOx Emission

By

S. Shiba
Okayama Prefectural University
Department of System Engineering
111 Kuboki, Soja, Okayama 719-1197 JAPAN

S. Nakaya, M. Tsue and M. Kono
University of Tokyo
Department of Aeronautics and Astronautics
7-3-1 Hongo, Bunkyou-ku, Tokyo 113-8656 JAPAN

Extended Abstract

Since 1970s, the environmental effects of future aircraft flying in the stratosphere are important topics with great interest. It is said that NOx in the ozone layer mainly affects the concentration of ozone. In the present study, the ozone layer was modeled in the chamber equipped with the ozone generator experimentally. NO and NO2 concentration were measured quantitatively by using both LIF methods in the quiescent mixed gas with photodissociation. NO and NO2 history data under several light conditions made the rates of photodissociation and other chemical reactions clear by comparing with numerical simulations. It is confirmed that the photochemical reactions in the ozone layer with NOx emission can be simulated in the chamber and by numerical simulation and the simulation can be expand to larger scale.

ISABE-2001-1050

Utilising Repair and Overhaul Experience in a Probabilistic Neural Network for Diagnosing Gas-Path Faults

R. Eustace[†] & P.C.W Frith[‡]
Defence Science and Technology Organisation,
Aeronautical & Maritime Research Laboratory, Australia

Abstract

This paper describes a diagnostic method that utilises experience gained during previous engine repair and overhaul maintenance to help identify the cause of gas-path faults requiring subsequent repair. The observation of causes and effects in previous repairs provides diagnostic information which is not encapsulated by traditional gas-path analysis methods. In this paper a Probabilistic Neural Network (PNN) uses such experience to build a database of knowledge. The catalogue of previous repairs is then used by the PNN to diagnose the cause of symptoms in current engines which require subsequent attention. The method presented has the potential to improve upon the traditional methods because it closely follows the approach taken by human experts. The method has been developed for application to the Pratt & Whitney TF30 engine, which is operated by the Royal Australian Air Force (RAAF) in the F-111 aircraft.

† *richard.eustace@dsto.defence.gov.au*
‡ *peter.frith@dsto.defence.gov.au*

ISABE-2001-1051

A MULTI-POINT GAS PATH ANALYSIS TOOL FOR GAS TURBINE ENGINES WITH A MODERATE LEVEL OF INSTRUMENTATION

Tomas U. J. Grönstedt
Chalmers University of Technology
Thermo and Fluid Dynamics
Göteborg, SE-412 96 Sweden
thgr@tfd.chalmers.se

Abstract

A method for estimating performance parameters in jet engines with limited instrumentation has been developed. The technique is applied on a non-linear steady state performance code using a range of operating points. A hybridized optimization tool using a real coded genetic algorithm to obtain an initial estimate of the performance parameters, and a gradient method to refine this estimate, has been implemented. The method is tested on a set of simulated data that would be available during performance testing of a PW100 engine. The technique has been successfully applied to the estimation of ten performance parameters using six simulated measurement signals. Several data sets ranging from three up to sixteen steady state points have been analyzed in order to determine the required range of data necessary for the estimation.

ISABE-2001-1053

Three-Dimensional Design and Optimisation of Turbomachinery Blades using the Navier-Stokes Equations

Shahrokh Shahpar
Aerothermal Methods, ML-83
Rolls-Royce plc
P.O. Box 31
Derby, UK

Extended Abstract

The Variable-Fidelity models of flow equations are used with different optimisation algorithms in a three-dimensional re-design and optimisation of a turbine nozzle guide vane. The low fidelity code is based on a linear sensitivity analysis tool, FAITH, whilst the high fidelity code solves the full three-dimensional Navier-Stokes equations. Performance and characteristics of various deterministic and stochastic optimisation techniques have been compared and discussed through a constrained minimization of the secondary-flow kinetic energy (SKE) in a transonic nozzle guide vane.

Based on the linear flow analysis, large reductions in the cost function of the order of 40-50% have been achieved using different optimisation routines. However, non-linear results show an increase of the order of 20-30% in cost function relative to the linearly predicted cost function. To address the non-linearity issue non-linear flow calculations were performed as part of the design search and optimisation. In order to avoid the large computational time usually associated with running 3D RANS codes a novel method was devised whereby the CFD flow field was initialized with the linear flow predictions followed by a limited number of CFD iterations. The part converged non-linear solutions were then used to estimate the objective functions and constraints. The non-linear optimisation strategy was successfully demonstrated to reduce the complex SKE cost function by as much as 60%.

Amongst the optimisers tested, the fastest one proved to be a code based on the Method of Feasible Direction (MFD). However, the MFD and a number of other local search methods failed in the presence of noisy part-converged CFD solutions. Alternatively, a variable step-size explorative code namely the Dynamic Hill climber and a stochastic code based on the Simulated Annealing methodology proved to be both robust and capable of significantly reducing the SKE whilst controlling the passage capacity.

Key words: *Aerodynamic Design and Optimisation; Comparison of heuristic, evolutionary and gradient-based optimisers; Variable-Fidelity (linear and non-linear) flow models.*

ISABE-2001-1055

Adjoint methods for turbomachinery design

M. S. Campobasso
M. C. Duta
M. B. Giles

Oxford University Computing Laboratory
Oxford, United Kingdom OX1 3QD

Abstract

This paper discusses the use of both steady and unsteady discrete adjoint methods for the esign of turbomachinery blades. Steady adjoint methods give the linear sensitivity of steady-state quantities such as the mass flow and the average exit flow angle to arbitrary changes in the geometry of the blades. This linear sensitivity information can then be used as part of a nonlinear optimisation procedure. The unsteady adjoint method is based on a single frequency of unsteadiness and gives the generalised force for a particular structural mode of vibration due to arbitrary incoming wakes. This can be used to tailor the radial variation in the incoming wakes to greatly reduce the level of forced vibration they induce.

The paper presents an overview of the discrete adjoint approach (which follows the work of Elliott and Anderson for external aerodynamic applications), explaining why it gives exactly the same results as linear perturbation methods, but at a greatly reduced computational cost. The key issues in the numerical implementation of the adjoint methods are discussed for both the Euler and the Reynolds-averaged Navier-Stokes equations. The correctness of the implementation is validated by comparison to both nonlinear and linear perturbation calculations.

Key words: blade design, adjoint methods, optimisation

ISABE-2001-1056

THE USE OF MULTIFIDELITY APPROXIMATIONS IN ENGINEERING DESIGN

Stephen Leary, Ivan Voutchkov, Atul Bhaskar and Andy Keane
The University of Southampton
Computational Engineering and Design Centre
School of Engineering Sciences, Highfield
Southampton, SO17 1BJ, UK

Extended Abstract

The problem of optimization using high cost models is common to many engineering design problems, design improvements are restricted by the computational burden involved, direct optimization being unrealistic.

To alleviate this burden, cheap approximations to the complex system are sought. Typical approximation methods make selective calls to this system using inputs from an experimental design. The methods typically employ curve fitting techniques, the expensive system response then being replaced by the approximation for the purposes of optimization.

One concern is the level of accuracy of the resulting approximation, as a result there has been a growing interest in the use of simple low fidelity models to the original expensive "high fidelity" model. The low fidelity model may use, for example, a coarse mesh in a finite element analysis whereas the high fidelity model may use a much finer mesh.

Combining the low fidelity model with the more accurate but expensive high fidelity models can provide a good combination of high accuracy and low cost, this is demonstrated on some simple finite elements models, results are encouraging.

Keywords: optimization, approximation methods

ISABE-2001-1058

Optimization of Three Dimensionally Designed Turbine Blades and Side Walls

Marc G. Nagel, Leonhard Fottner
Institut für Strahlantriebe
Universität der Bundeswehr München
D-85577 Neubiberg, Germany

Ralf-D. Baier
MTU Aero Engines
D-80995 München

Abstract

This paper presents a system for the aerodynamic design optimization of three dimensionally defined turbine blades and side walls. The system consists of a fully automatic three dimensional design process performed successively with a numerical optimization algorithm. The base structure of the system containing the modules axial side wall parametrisation, blade profile parametrisation, non-axisymmetrical end wall contouring, grid generation, flow solving using a Navier–Stokes solver, postprocessing and the numerical optimization algorithm is described in detail. By defining a target function and various boundary conditions, a blade and side wall design can be performed simultaneously. The potential of the system is demonstrated by an optimized redesign of a low pressure turbine cascade to reduce the secondary flows and the integrated losses.

ISABE-2001-1062

A Vaneless Counter-Rotating Turbine Design Towards Limit of Specific Work Ratio

By

Lucheng Ji Xiaobo Quan* Wei Li Jianzhong Xu
Institute of Engineering Thermophysics, Chinese Academy of Sciences
P.O. Box 2706
BeiJing 100080, China
E-mail: jlucheng@yahoo.com or jlc@mail.etp.ac.cn

*Xi'an Jiao Tong University

Abstract

To cater to the needs for the aircraft engine with higher Thrust/Weight ratio, the vaneless counter-rotating turbine (hereafter refers as VCRT) is becoming especially preferred. But the design of a practicable VCRT seems to be faced with great difficulties. In the present investigation preliminary theoretical analysis about work capability of VCRT is firstly carried out. Though the analysis is somewhat idealistic, it does provide guidance for the development of VCRT. The studies indicate supersonic exit flow and higher flow angle at the outlet of high pressure turbine (HPT) stage is needed for the use of VCRT under the current engine cycle arrangements. Appropriate modification of the cycle parameters may alleviate the difficulties involved in aerodynamic design of VCRT.

ISABE-2001-1064

STAGED COMBUSTION CONTROL SYSTEMS FOR APPLICATION TO AEROSPACE GAS TURBINE ENGINES

TV Breikin, S Regunath, SM Hargrave, HA Thompson,

PJ Fleming and S Riley*

Rolls-Royce University Technology Centre, Department of Automatic Control & Systems Engineering, University of Sheffield, UK

*** Team Leader, Controls Technology Acquisition, Rolls-Royce plc, Derby, UK**

1. Abstract

The current paper outlines the motivation for developing a staged combustion system and highlights the control-related issues of such an approach for civil aerospace application. It discusses the findings of an analysis of the fuel distribution control system to address issues of optimal fuel distribution and bumpless transfer during burner priming and purging. An overview of the control schemes to be employed in the distribution controller of the research programme is presented.

Environmental considerations are driving the development of low emission gas turbine engines for civil aerospace applications. Emissions control requirements for civil aircraft are set by the International Civil Aviation Organisation (ICAO). Limits were originally set in terms of pollutants produced per unit thrust or unit mass of fuel burned over Landing and Take-Off cycle. Tightening of the NO_x standards have taken place and additional emphasis has been placed on CO emission levels following a recognition of the impact of emissions generated at cruise operating conditions. The tightening of ICAO regulations is approaching the limit of conventional combustion chamber design.

Staging the combustion between combustion zones of different geometry offers the potential to reduce the overall emissions of a gas turbine engine by enabling a more efficient burn over a wider operating range than a single zone combustor design is able to achieve. Its aim is to reduce NO_x while maintaining other pollutants at low levels by introducing combustion burners sequentially. This is achieved by sharing the fuel load among several fuel and air stages, thus enabling the control of the local Fuel-to-Air Ratios (FAR) to minimise NO_x formation rates as the power increases.

The primary function of the fuel distribution control system is to allocate the demanded fuel flow between the banks of burners within the pilot and main combustion zones to minimise the overall engine emissions, whilst maintaining the engine within required operating constraints. Three strategies were proposed for the control of the fuel distribution: open-loop scheduling, closed-loop control and closed-loop trim control. Practical issues of hysteresis and bumpless transfer compensation in the priming and purging of the main burner sets are being addressed to prevent hunting between lit burner configurations and minimise thrust disturbances respectively.

The research project is entitled LECCAGT (Low Emissions Combustion Control for Aero Gas Turbines) and is a UK Department of Trade and Industry funded Foresight Link programme. It is lead by TRW Aeronautical Systems in collaboration with Rolls-Royce plc and the Universities of Sheffield and Cranfield to rig demonstrate the fuel metering of a staged combustion system for civil aerospace applications.

2.

ISABE-2001-1065

ASSESSMENT OF GAS TURBINE COMBUSTOR PERFORMANCE AND EMISSIONS USING A WELL-STIRRED REACTOR

Dilip R. Ballal[#] and Joseph Zelina[*]

[#]University of Dayton, Dayton, OH 45469 U.S.A.
[*]Air Force Research Laboratory, Wright-Patterson Air Force Base, OH, U.S.A.

Abstract

The design and development of low-emissions, lean premixed aero gas turbine combustor is very challenging because it entails many compromises. To satisfy the projected CO and NO_x emissions regulations without relaxing the conflicting requirements of combustion stability, efficiency, pattern factor, and relight capability demands great design ingenuity. The well-stirred reactor (WSR) provides a laboratory idealization of an efficient and highly compact advanced combustion system of the future. In this paper, we have studied the combustion performance and emissions using a toroidal WSR.

It was found that a simple analysis based upon WSR theory provided good predictions of the WSR lean blowout limits. The WSR combustion efficiency was 99 percent over a wide range of mixture ratios and reactor loading. CO emissions reached a minimum at a flame temperature of 1600K and NO_x increased rapidly with an increase in flame temperature, moderately with increasing residence time, and peaked at or slightly on the fuel-lean side of the stoichiometric equivalence ratio.

Detailed kinetic modeling was applied to predict emissions CO and NO_x emissions. The temperature for minimum CO emission decreases with increasing fuel carbon number, a trend accurately captured by the model. Predictions of NO_X are in reasonably good agreement with measurements for various fuels *without* introducing adjustments to kinetic rate parameters. Finally, emissions maps of different combustors were plotted and showed that the WSR has the characteristics of an idealized high efficiency, low emissions combustor of the future.

ISABE-2001-1066

NUMERCIAL SIMULATION OF CONFINED KEROSENE SPRAY FLAMES AT ELEVATED PRESSURE

By

D. LEPORT, S. A. L. PERERA and J. B. MOSS*
School of Mechanical Engineering
Cranfield University
Cranfield, Bedford MK43 0AL
U.K.

Extended Abstract

Whilst significant progress has been made in the numerical simulation of increasingly complex gas turbine combustor flowfields, and including thermal NO_x emissions, the prediction of smoke and its coupling with fuel-air mixture preparation and radiation heat transfer remains a challenging problem. The kerosene spray flame introduces a variety of physical and chemical processes that are both incompletely understood, for example, those of kerosene pyrolysis chemistry and soot formation, and imperfectly modelled, as of turbulent mixing within the liquid fuelled spray. Detailed investigations of sprays have been reported, incorporating PDPA characterisation of the liquid and gas phases, but in general these studies are undertaken under near-isothermal conditions and are restricted to measurements at atmospheric pressure. Fisher and Moss (1998) performed and experimental study and detailed property maps were constructed for gas phase mixture fraction, temperature, soot volume fraction and radiative flux in a confined kerosene spray flame, burning at pressures up to 12 bar across a range of air-assisted injector AFRs. These data have recently been supplemented by measurements of fuel droplet size and velocity using PDPA at pressures up to 6 bar and for injector AFRs in the range of 1.9-5.7 (Leport et al; 1999, Leport; 2001). These experiments were undertaken specifically to provide data for CFD model development and evaluation. Though air-assited, the injector configuration was chosen to be non-swirling, in principle, to focus on the fuel related issues. The present paper describes this model development and assesses the sensitivity of smoke predictions to details of the representation of the processes of droplet transport, evaporation, mixing and combustion.

The CFD modelling was undertaken within the framework provided by both a commercially available code (FLUENT) and an in-house research code (SOFIE). The turbulent dispersion of fuel droplets and the heat/mass transfer between them and the gas phase is modelled through a Lagrangian droplet tracking approach. Turbulence interaction between the flowfield and combustion chemistry is accommodated using the laminar flamelet model. For purposes of chemical kinetic modelling kerosene is considered to comprise a blend of 80% undecane and 20% benzene, but has only a single boiling point in respect of evaporation. A library of laminar flamelets has been constructed for kerosene-air mixtures, burning under the experimental conditions of pressure and preheated air temperature, following the strategy proposed by Brocklehurst et al (1998). The process of soot formation is represented

* Corresponding author: j.b.moss@cranfield.ac.uk

by the two-equation model, proposed by Brookes and Moss (1999), with the soot precursor species identified as acetylene and benzene. Radiative exchange is computed using the discrete transfer model.

A critical issue in the numerical simulations is that of defining the injector boundary condition. Although measurements have been made of both fuel droplet properties and the gas phase mixture fraction, uncertainties are introduced into the partitioning of the fuel fluxes. PDPA measurements of velocity of the smallest fuel droplets, for example, may be unrepresentative of the gas flow in the near field of the injector and the integrated measurement of spatially resolved droplet size and velocity does not accurately reproduce the liquid flow rate. The sensitivity of the calculations to the prescription of boundary conditions and alternative physical model is explored in the full paper.

An illustrative solution at a pressure of 2 bar and injector AFR of 2.85 is summarised in the accompanying figures. Predictions of the gas phase mixture fraction, temperature and soot volume fraction are compared with experimental measurement. The rates of mixing inferred from the measurements are generally more rapid than is implied by the simulations and the early flame development is inhibited. Temperature and soot concentration levels are plausibly reproduced.

References

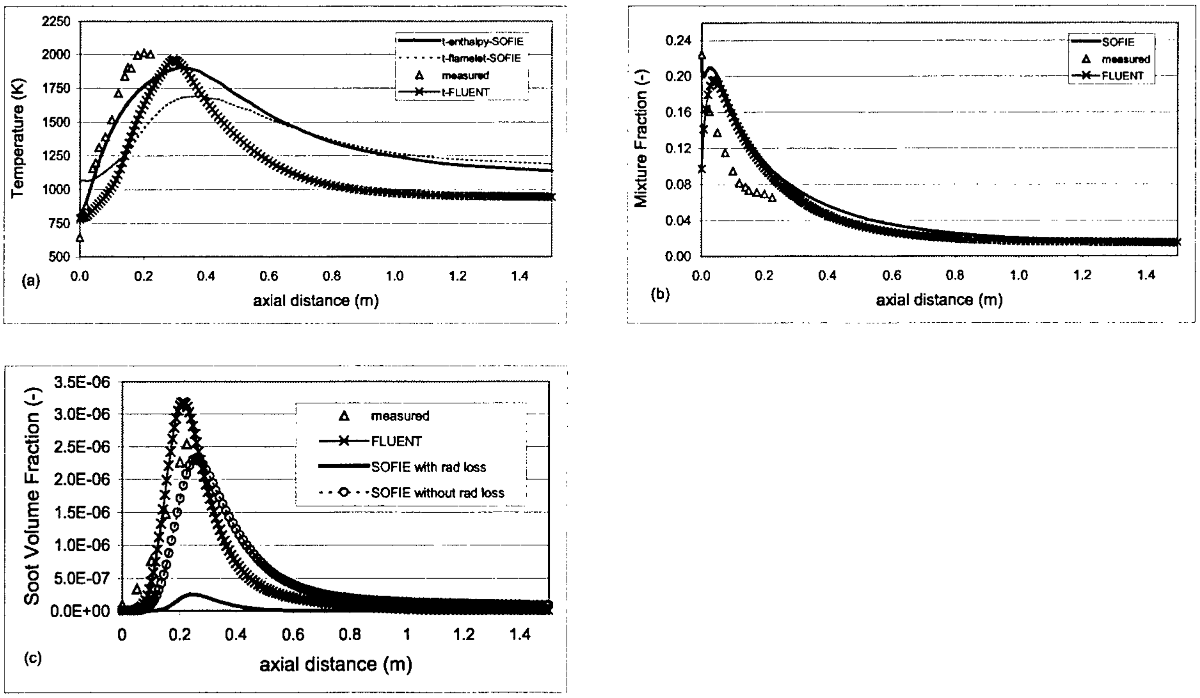
Brocklehurst, H. T., Hurley, C. D., Moss, J. B. and Priddin, C. H. (1998), Soot and Radiation Modelling in Gas Turbine Combustors, NATO AVT Mtg. *Gas Turbine Engine Combustion, Emissions and Alternative Fuels*, RTO-MP-14.

Brookes, S. J. and Moss, J. B. (1999) Predictions of Soot and Thermal Radiation Properties in Confined Turbulent Jet Diffusion Flames, *Combustion and Flame*, **116**, 486-503.

Fischer B. A., Moss J. B., (1998) The Influence of Pressure on Soot Production and Radiation in Turbulent Kerosene Spray Flames, *Combustion Science and Technology*, **138**, 43-61.

Leport, D., Jasuja, A. K. and Moss, J. B. (1998) PDPA Measurements in a Combusting Kerosene Spray, *Fifteenth Annual Conference on Liquid Atomisation and Spray Systems*, Toulouse, France.

Leport, D. (2001) Modelling Kerosene Spray Flames at Elevated Pressure, *MPhil Thesis*, Cranfield University (in preparation).


2250
2000
1750
1500
1250
1000
750
500
Temperature (K)
(a)
0.0
0.2
0.4
0.6
0.8
1.0
1.2
1.4
axial distance (m)
t-enthalpy-SOFIE
t-flamelet-SOFIE
measured
t-FLUENT
0.24
0.20
0.16
0.12
0.08
0.04
0.00
Mixture Fraction (-)
(b)
SOFIE
measured
FLUENT
3.5E-06
3.0E-06
2.5E-06
2.0E-06
1.5E-06
1.0E-06
5.0E-07
0.0E+00
Soot Volume Fraction (-)
0
1
(c)
FLUENT
SOFIE with rad loss
SOFIE without rad loss

ISABE-2001-1067

PREDICTING PERFORMANCE FOR ALTITUDE RE-IGNITION

Véronique QUINTILLA[1], Michel CAZALENS[1], Renaud LECOURT[2], Gérard LAVERGNE[2]

[1] SNECMA
Site de Villaroche
Département Aérocombustion
77550 Moissy-Cramayel
France

[2] ONERA-Centre de Toulouse
DMAE
2, avenue Edouard Belin
31055 Toulouse cedex 4
France

ABSTRACT

During flight, many events such as ingestion of ice, water or exhaust gas can induce extinction. The design of the combustor must then ensure that it will be possible to restart the engine inside a given domain of Mach and altitude. In this context, experiments have been performed, and a time dependant 0-dimensional model has been developed to predict what occurs to a cluster composed of fuel droplets, when it is submitted to the spark inside the combustion chamber. The complete procedure of calculation where cluster model is coupled with NS3D computation is applied to the study of the optimal igniter position for a real combustor. First results are presented.

ISABE-2001-1069

How to Resolve the Precooler Icing Problem for Combined Cycles of the KLIN Type

Vladimir Balepin, MSE Technology Applications, Inc., Butte, MT, USA
Don Bai, NASA Marshall Space Flight Center, Huntsville, AL, USA

This paper discusses a problem of the operation of precooled and liquefaction cycles for high speed propulsion that involves ice buildup on the air side of the heat exchanger surface due to atmospheric moisture and possible performance deterioration. Both speed and altitude limits based upon the theoretical consideration of ice formation have been generated for specific Cape Canaveral climate conditions. These results have been confirmed by previous model testing and engine testing. The method of icing prevention before reaching these limits applicable to KLIN Cycle and its derivatives is discussed.

ISABE-2001-1070

PERFORMANCE EVALUATION OF THE NASA GTX RBCC FLOWPATH

Scott R. Thomas, Donald T. Palac, Charles J. Trefny, and Joseph M. Roche
National Aeronautics and Space Administration
Glenn Research Center
Cleveland, Ohio 44135

Abstract

The NASA Glenn Research Center serves as NASA's lead center for aeropropulsion. Several programs are underway to explore revolutionary airbreathing propulsion systems in response to the challenge of reducing the cost of space transportation. Concepts being investigated include rocket-based combined cycle (RBCC), pulse detonation wave, and turbine-based combined cycle (TBCC) engines.

The GTX concept is a vertical launched, horizontal landing, single stage to orbit (SSTO) vehicle utilizing RBCC engines. The propulsion pod has a nearly half-axisymmetric flowpath that incorporates a rocket and ram-scramjet. The engine system operates from lift-off up to above Mach 10, at which point the airbreathing engine flowpath is closed off, and the rocket alone powers the vehicle to orbit.

The paper presents an overview of the research efforts supporting the development of this RBCC propulsion system. The experimental efforts of this program consist of a series of test rigs. Each rig is focused on development and optimization of the flowpath over a specific operating mode of the engine. These rigs collectively establish propulsion system performance over all modes of operation, therefore, covering the entire speed range.

Computational Fluid Mechanics (CFD) analysis is an important element of the GTX propulsion system development and validation. These efforts guide experiments and flowpath design, provide insight into experimental data, and extend results to conditions and scales not achievable in ground test facilities. Some examples of important CFD results are presented.

Keywords: Combined cycle propulsion; Hypersonic propulsion; Ramjet; Scramjet; Air-breathing launch vehicle; Single stage to orbit

ISABE-2001-1071

Hyper-X Program Status

C. R. McClinton and R. Voland
NASA Langley Research Center
Hampton, VA
USA

ABSTRACT

This paper provides an overview of NASA's Hyper-X program, which is moving hypersonic, airbreathing vehicle technology from the laboratory to the flight environment. Hyper-X focus is the 12-foot long X-43 scramjet powered vehicle. Technology development includes X-43 risk reduction, design method refinement and flight validation, and planning for the future. The first Mach 7 flight of the X-43 occurred June 2, 2001. Problems encountered in the boost phase precluded reaching the scramjet test condition, but extensive data generated is being studied to verify all systems. The second flight is currently scheduled for late in CY 2001. When flown, this will be the first flight of an airframe-integrated scramjet-powered vehicle.

Key Words:

Launch Vehicles, Scramjet, Flight Tests, Hypersonic Technology

ISOABE-2001- 1073

French Activities In Hypersonic Airbreathing Propulsion Status and Perspectives

François FALEMPIN – EADS Aerospatiale Matra Missiles – Chatillon – France
francois.falempin@missiles.aeromatra.com

Abstract

After the end of the National PREPHA program, EADS-AMM and ONERA have taken the initiative in starting further works to preserve the intellectual and material investment and to improve mastery of hypersonic airbreathing propulsion.

ONERA and DLR decided to join their efforts in the common research program JAPHAR, which aims at developing a hydrogen fuelled dual mode ramjet for the Mach number range from 4 to 8. ONERA is also working with SNECMA and PRATT & WHITNEY for developing a fuel-cooled composite material structure technology.

EADS-AMM leads a cooperation with Moscow Aviation Institute to develop a dual-mode dual fuel ramjet, operating from Mach 3 to Mach 12 with a variable geometry. EADS-AMM and EADS-LV are also developing an innovative technology for fuel-cooled composite material structures.

Finally, under the aegis of French MoD, EADS-AMM and ONERA are leading the PROMETHEE R&D program to improve knowledge on hydrocarbon fuelled dual mode ramjet for missile application. Its aims at developing a propulsion system able to power a missile from Mach 2 to Mach 8. It is expected to end early in 2002.

At this time, the development of a minimum autonomous experimental vehicle could be decided to study the "thrust minus drag" balance in the Mach number range from 4 to 8.

ISABE-2001-1074

Engine Development for Space Access: Past, Present and Future

McClinton, C.R.; Andrews, E.H.; and Hunt, J.L.
NASA LaRC, Hampton, VA, USA

Abstract

Efficient airbreathing engines for reliable, affordable space access have been studied in the USA for over 40 years. The heart of these systems, the scramjet engine, has been investigated continuously during this time, with over 4,000 tests performed on at least 40 scramjet engines. Efforts to integrate the scramjet with rockets for higher speed and air-augmented rockets or turbojets for lower speed have been investigated continuously since 1984. This paper presents overview of accomplishments of the USA efforts. Additional research and technology development requirements are discussed, and a plan presented to complete this technology development.

Key Words: hypersonic, propulsion, scramjet, plans

ISABE-2001-1075

RESEARCH ACTIVITIES ON SCRAMJETS AT NAL-KRC IN JAPAN

By

Nobuo Chinzei
National Aerospace Laboratory
Kakuda Space Propulsion Laboratory
Kakuda, Miyagi, 981-1525
Japan

Extended Abstract

Research activities on scramjet engines at the National Aerospace Laboratory, Kakuda Research Center (NAL-KRC) are reviewed from the initial component tests through the engine tests in its Ramjet Engine Test Facility. Through these activities, many valuable materials for designing the scramjet engine have been obtained. Future prospects based on those results are also described.

The initial research activities on components began in the late 1970's for the combustor as the first subject followed by other component studies. Based on their results, a side-wall-compression type scramjet engine was designed and fabricated in 1991 through 1994 to investigate the interaction among the components and its overall performance.

In parallel with the design and fabrication of the engine, NAL-KRC built a free-jet-type hypersonic propulsion wind tunnel (RJTF) in 1994 and a free-piston high-enthalpy shock tunnel (HIEST) in 1997, respectively. With the RJTF, scramjet engines can be tested in the range of simulated flight Mach numbers of 4, 6 and 8. In the HIEST, it is possible to test a scramjet engine at higher Mach number and hypersonic aerodynamic phenomena. Its stagnation enthalpy and pressure are up to 25 MJ/kg and 150 MPa, respectively.

The engine model is based on one of the modules of a hypothetical airframe-integrated engine (Fig. 1). It consists of six major components, namely, an inlet leading edge, a downstream part of the inlet, a fuel injector, and diverging sections of a combustor and a nozzle. They are replaceable for parametric tests by independently changing their geometry. Most of its dimensions are based on the results of the component tests conducted at the NAL-KRC. Overall length is 2100 mm with equal entrance and exit dimensions of 200 mm in width and 250 mm in height.

Approximately 160 tests have been conducted for the engine model with the RJTF. The first engine test was done under M4 condition in March 1994. Over the past four years, through experience gained in the test series and analysis of data, various improvements in testing techniques and data reliability have been successfully attained.

Figures 2 and 3 show a thrust increment, *dF*, i.e., the difference between the axial component of the Force Measuring System outputs with and without fuel injection, against fuel equivalence ratio, *phi* for flight conditions of Mach 4 and 6, respectively. The thrust begins to rise steeply beyond a certain value of *phi*'s, which we defined as a transiton from the "weak" to the "intensive mode." Further increase in *phi* causes the engine to fall into unstart.

Figure 2 compares the effect of the isolator length in the Mach 4 tests, clarifying that a combustor/inlet interaction was mostly eliminated by employing a long isolator. For the Mach 6 condition shown in Fig. 3, there was no difference in the the thresholds value of the fuel flow rate for the mode transition, but the long isolator model falls to unstart at higher *phi*'s because of relaxation of the combustor/inlet interaction

After several series of tests for the flight conditions of Mach 8 with the above-mentioned first generation model, which showed unsatisfactory results., the second-generation engine was designed and fabricated. It was tested in March, 2000 for the first time, and the follow-on series conducted in March through April, 2001 have completed recently. The results for the Mach 8 tests are shown in Fig. 4 for the first and second generation engine. The thrust performance have been gradually improved during the process of seeking for the better thrust generation including the second generation model.

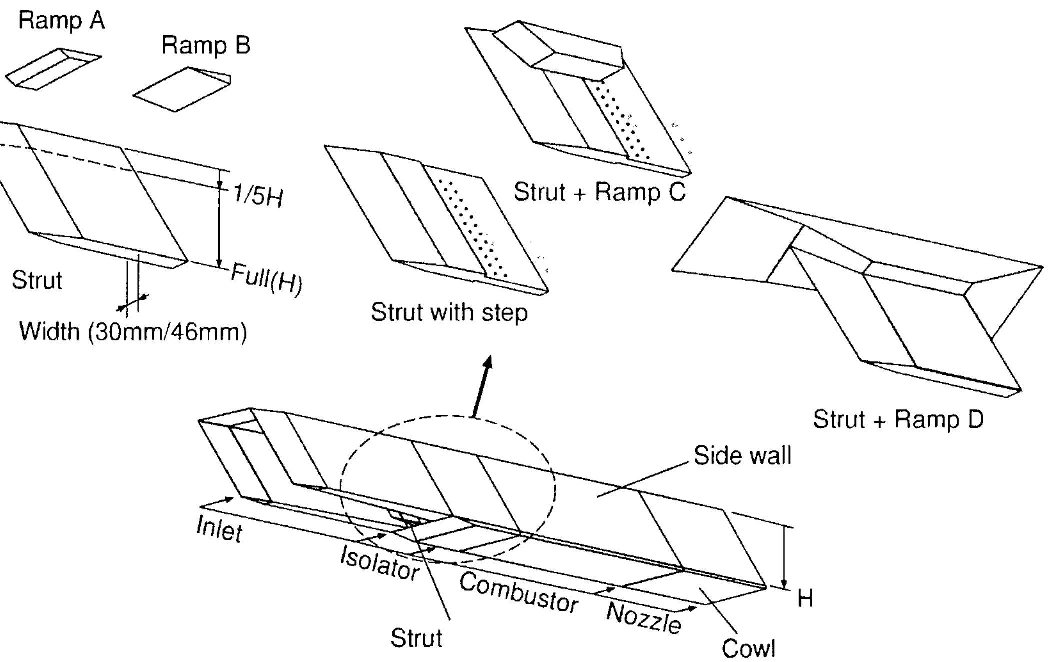


Fig. 1 Scramjet engine model tested

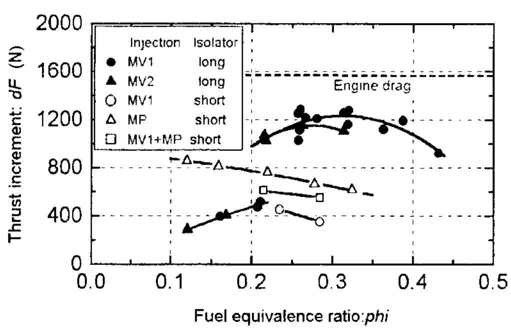


Fig. 2 Thrust increment found in M4 tests

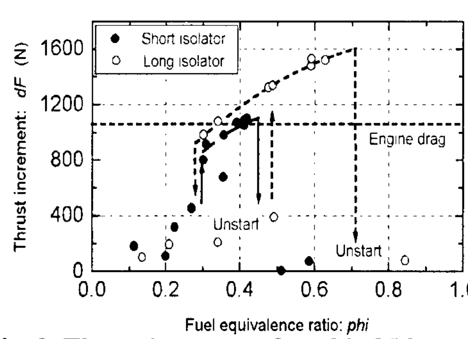


Fig. 3 Thrust increment found in M6 tests

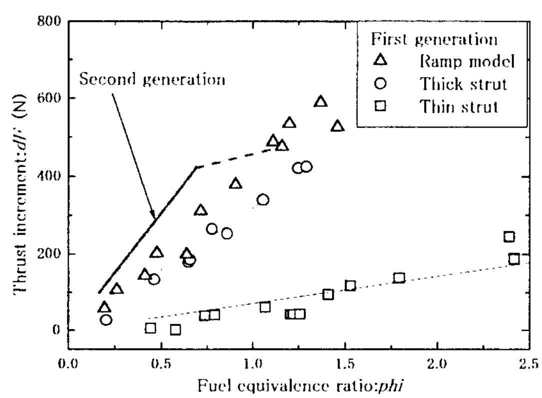


Fig. 2 Thrust derived under M8 condition

ISOABE - 2001 - 1078

The French PROMETHEE program on hydrocarbon fueled dual mode ramjet
Status in 2001

Laurent SERRE – ONERA – Palaiseau - France
François FALEMPIN – EADS-Aerospatiale Matra Missiles – Châtillon - France

Abstract

This paper provides detailed description of the French program PROMETHEE on hydrocarbon fueled dual mode ramjet, which was started at the beginning of 1999 by the French government. After replacing this program into the context of the other programs related to hypersonic airbreathing propulsion, the technical goals are described, and the results obtained after two years are presented.

During the first year, the definition of a generic missile provided specifications for the propulsion system. Several concepts have been studied and compared using performance simulation on flight trajectories. The selected concept is a variable geometry dual mode ramjet equipped with a thermal throat for the subsonic combustion phase. A full scale of the engine was defined for a first evaluation of the combustion performances, which is going to start by the end of 2001 with connected pipe tests. In a fist step, the air inlet will be tested separately, and then will be coupled to the engine for semi-free jet testing beyond the first phase of the present program.

In parallel with this experimental approach, the design of an operational propulsion system is in progress, with a special emphasis on the fuel circuit and its use as a cooling system for the structures of the combustion chamber. The problems associated with the fuel decomposition in very high heat load environment are examined, together with the associated technologies.

Finally, the first elements concerning the current reflection about the future needs of technological developments and ground demonstration are presented.

ISABE-2001-1079

UNSTEADY BLADE PRESSURE DISTRIBUTIONS ON A COUNTERROTATING PROPFAN AT ON- AND OFFDESIGN CONDITIONS

by

R. Schnell; L. Wallscheid

German Aerospace Center DLR

Institute of Propulsion Technology

Linder Höhe, 51147 Köln

Fon: (0049)- 2203 601 2000; Fax (0049)-2203 64395

Email:

rainer.schnell@dlr.de
lothar.wallscheid@dlr.de

Extended Abstract:

In order to investigate the nature of those phenomena causing pressure fluctuations of varying intensity on compressor blade surfaces, numerical studies have been performed on a counterrotating propfan at on- and off design conditions. Therefore, the unsteady flow field of the propfan CRISP (Counter **R**otating **I**ntegrated **S**hrouded **P**ropfan, a concept study by german engine manufacturer MTU for future UHBR engines) has been investigated numerically and experimentally. The design and one off design point at approx. 50% speed have been investigated using the unsteady Navier-Stokes code TRACE-U [4]. Due to the favourable nature of the 3D-flowfield and to reduce the numerical effort, the calculations were carried out on a cylindrical slice of constant radius. At design point conditions the flow field is characterised by large transonic areas in the relative frame of reference of both rotors while being in the subsonic range for the off-design case. To validate the numerical calculations, experimental investigations have been performed on a 1m-model at the compressor test facility (M2VP) at DLR-Cologne providing a large amount of experimental data (3D-L2F, unsteady pressure measurements on the blade surface and behind the second rotor etc.). The experimental results are topic of numerous publications [1,3].

The main difference between the two operating conditions is the existence of shocks in both rotors at design point compared to the subsonic flow field at 50% speed. At the design point a strong upstream effect of the second rotor upon the first was found. The reason for this is the impingement of the rotor-II shock on the aft section of the rotor-I blade profiles. Almost no dynamic portions of the blade pressure could be found in the front region of rotor-I. The supersonic region with the nearly normal shock in the rotor-I blade passage prevents pressure fluctuations to move upstream. Both shocks show a very stable behaviour. Regarding the second rotor one of the main causes for pressure fluctuations is the impingement of the rotor-I wake. Additionally, pressure waves due to reflections induced by the upstream rotor II-rotor I interaction could be recognised on the blade surface of the rotor-II which leads to a fourth order pressure peak.

At the investigated off-design point no significant upstream influence from the second rotor upon the first could be found. The main sources causing pressure fluctuations on the blade surface were found as wake shedding of the first rotor, pressure waves emanating from the leading edge of the second rotor and impinging on adjacent blade rows and reflections of these pressure perturbations running upstream. The dominant phenomenon for pressure unsteadiness on the second rotor blades was the wake shed by the first rotor. The negative jet effect as well as the typical vortices surrounding the rotor-I wake were well resolved by the numerical code and its influence upon blade pressure of the second rotor could be studied in detail. Though pressure fluctuations were small at this operating point the phenomena causing these fluctuations were evident and thus its influence upon blade surface pressure could be studied in detail.

References:

[1] L. Wallscheid, „*Phänomenologische Untersuchung der zeitabhängigen Strömung in einem gegenläufigen Propfan*", DLR Forschungsbericht 1999-24

[2] L. Wallscheid; F. Eulitz; R. Heinecke, *„Investigation of Unsteady Flow Phenomena in a Counterrotating Ducted Propfan"*, ASME 98-GT-251

[3] P. Schimming; S. Schmitt; R. Schnell et. al., *"Experimentelle Untersuchungen zur Aeroakustik, -elastik und –dynamik am CRISP 1m-Modell"*, Abschlussbericht Enginge 3E, Förderkennzeichen LFT 9601

[4] F.Eulitz; K. Engel; D. Nürnberger; S. Schmitt; K. Yamamoto, *„On Recent Advances of a Parallel Time-Accurate Navier-Stokes-Solver for Unsteady Turbomachinery Flow"*, 4th ECCOMAS CFD-Conference, Athens, Greece

	Design	**Off-Design**
Corrected Mass Flow	166 kg/s	108 kg/s
Overall Total Pressure Ratio	1.24	1.05
Inlet abs. Mach Number	0.77	0.41
RPM Rotor I	4980 1/min	2600 1/min
RPM Rotor II	4316 1/min	2340 1/min
Outer Diameter	1m	
Number Of Blades Rotor I/Rotor II	10 / 12	

Table 1: Flow Parameters of the two different investigated Operating Points

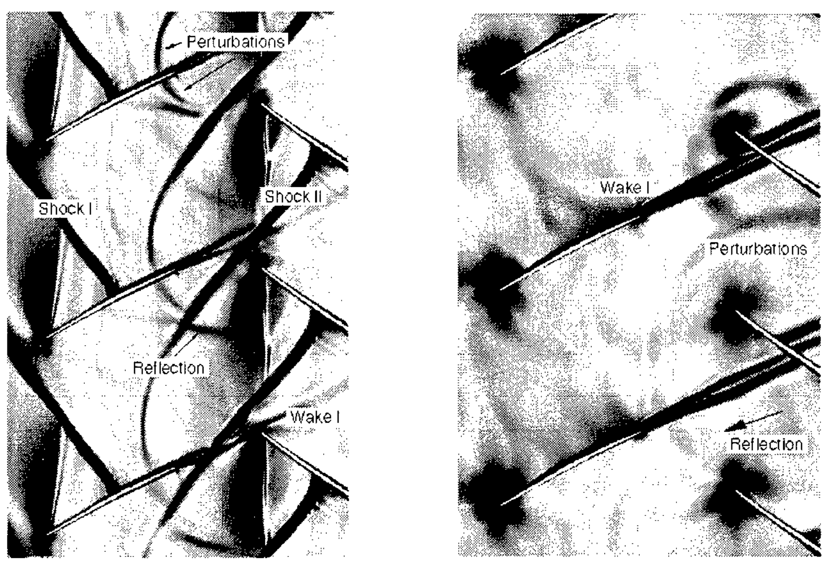


Figure 1: „Numerical Schlieren" Picture (Density Gradients) for both Operating Points; the Main Flow Phenomena causing Pressure Fluctuations on Blade Surface are denoted;

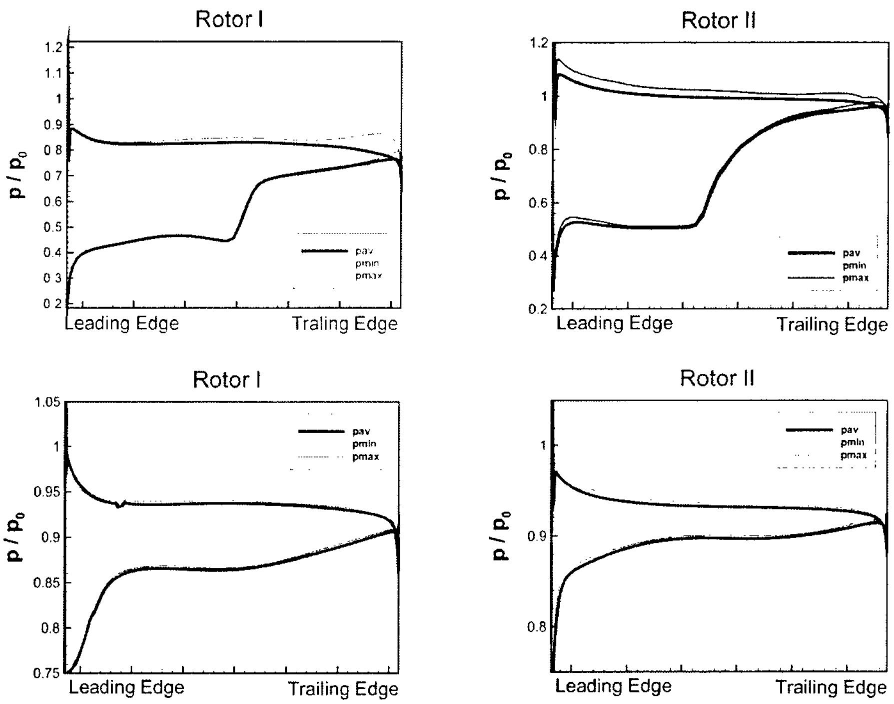


Figure 2: Blade Pressure Distribuation of both Rotors at **Design (above)** and **Off-Design** Conditions (**underneath**); At each axial Position the Time-Average, the Maximun and Minimum Blade Pressure is plotted

ISABE-2001-1081

RECENT PROGRESS IN THE NUMERICAL SIMULATION OF UNSTEADTY VISCOUS MULTISTAGE TURBOMACHINERY FLOW

F. Eulitz, D. Nürnberger, S. Schmitt, A. Zachcial
German Aerospace Center (DLR) – Institute of Propulsion Technology
Köln, Germany

Abstract

The current paper will report on recent important steps in the development of a CFD solver towards a more realistic simulation of turbomachinery flow.

Blade row interaction effects in multistage-turbomachinery cannot be neglected in a lot of applications as they influence efficiency and operating point. The current method was from the beginning devoted to time-accurate simulations of multiple blade-row configurations avoiding questionable coupling of single-blade-row-calculations.

One of the most important points is the change from idealized geomtries used in the past towards full consideration of the real geometry including cavities. Cavity flows interact with the main flow and significantly impact the efficiency and operating range. The developed approach decouples the generation of the structured meshes of blade passage and cavities. The resulting unequal grid surfaces at the interface between both meshes are coupled by a chimera technique. The accuracy of the coupling will be demonstrated with an example.

Of major importance to an accurate loss estimation is the correct invokement of laminar-to-turbulent boundary layer transition. In unsteady environments with multiple blade rows wake-induced transition will lead to a time-dependent transition behaviour. The first approach that will be presented is within the framework of unsteady Reynolds-Averaged Navier-Stokes simulations the combination of a progressive one-equation turbulence model with a transition criterion. The modeling was especially adapted to unsteady turbomachinery flows and is capable to correctly resolve unsteady wake-induced transition what will be proved with an example. The model will be used to investigate a 2.5-stage low-pressure-turbine module with multiple wake-blade interactions.

The second approach will extend the investigations on boundary layer transition by migrating from a Reynolds-Averaged approach to a Large-Eddy-Simulation. The first 1.5-stages of the same low-pressure-turbine are resolved now using 10 mio. grid points making the simulation a great challenge even to todayest modernst parallel supercomputers.

The increase in physical resolution requested by the above mentioned tasks even superceeds the pronounced increase in computing performance available today. The development of faster, highly-accurate time integration methods is therefore essential. A newly developed scheme cuts down the run time of time-accurate simulations down to 10% of the method that has been used before. Accuracy and acceleration will be demonstrated by an example.

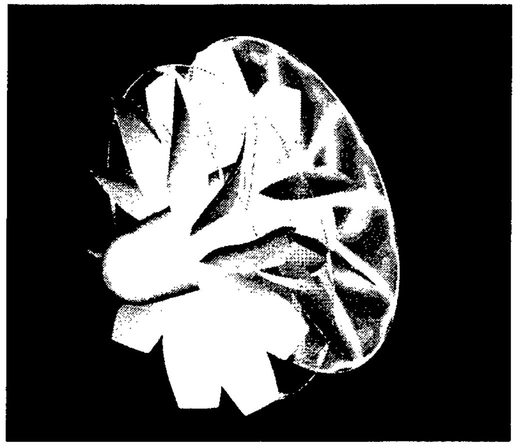


Figure 1: Interaction of wake and tip-vortex in a counter-rotating propfan.

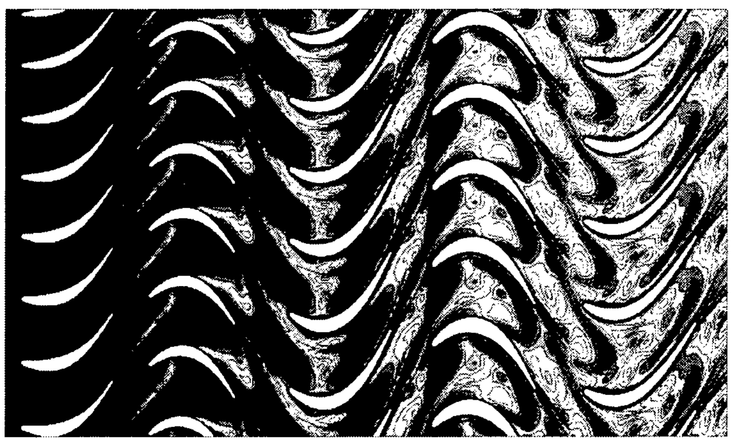


Figure 2: Multiple blade-row interaction in a 2.5 stage low pressure turbine.

ISABE-2001-1082

PARTICLE IMAGE VELOCIMETRY CHARACTERIZATION OF SECONDARY FLOW EFFECTS IN A TURBINE STAGE INCLUDING THE EFFECTS OF UNSTEADY FORCING FUNCTIONS

Kirk D. Gallier, Patrick B. Lawless and Sanford Fleeter
Maurice J. Zucrow Laboratories
Purdue University
West Lafayette, Indiana

Abstract

Recent work at Purdue University has focused on the nature of complex secondary flows in turbines which have a strong impact on external heat transfer. The most important of these are the effects on seal purge air on rotor platform flow development, and the nature of tip clearance flows. To describe these flows, experiments have been performed on the Purdue Research Turbine, a two-stage cold flow facility that allows the contribution of blade row interactions in the development of these flows to be observed. The current paper focuses on the hub seal portion of this work.

Keyword: Secondary Turbine Flows PIV

ISABE-2001-1084

TOWARDS LONG LENGTH SCALE UNSTEADY MODELLING IN TURBOMACHINES

By

L Xu, T P Hynes and J D Denton
Whittle Laboratory
University of Cambridge

Extended Abstract

The modelling issue of the long length scale unsteadiness in turbomachines is discussed in this paper and a new three dimensional CFD method with a hierarchy of bodyforce models has been proposed. A program has been developed to deal with the long length scale problem using efficient large mesh sizes and large time steps together with bodyforce models. In the method proposed, different densities of meshes can be used to resolve desired flow features whilst fine structures are modelled by simulated bodyforces. In the middle of the hierarchy, a distributed viscous bodyforce model is employed which can be either calculated using a very simple viscous wall shear stress model or extracted from a 'stand-alone' proper three dimensional single passage calculation using fine mesh. By avoiding devoting computing efforts to fine viscous scales the proposed method is several orders of magnitude more efficient compared with the 'standard' fine mesh N-S unsteady calculations while maintaining respectable resolution down to the scale of blade to blade variation, including wake/potential interactions between blade rows.

Several cases of unsteady flows including complex flows through a low speed multistage turbine and distorted inlet flow through a highly loaded transonic fan are studied using the proposed model. Comparisons with the results from a fine mesh N-S calculation are made where possible. The results show that the proposed model is able to capture all prominent features of the unsteady flow in the cases studied and the results are in good agreement with the fine mesh N-S calculation both qualitatively and quantitatively. The calculation with a fine mesh N-S solver would take weeks even months to complete whilst for the proposed model it takes merely few hours.

key words: unsteady turbomachinery flow calculations; bodyforce model; non-axisymmetric turbomachinery flow

ISABE-2001-1086

A SYNOPSIS OF GAS TURBINE COMBUSTOR DESIGN METHODOLOGY EVOLUTION OF LAST 25 YEARS

Hukam C. Mongia
GE Aircraft Engines, Cincinnati, Ohio, U.S.A.

Abstract

The gas turbine combstor design approach has changed radically during the last 25 years from being based essentially on empiricism to setting highly aggressive goals for the next ten years: Design the combustion system analytically and test to verify the design intent. The design methodology has progressively gone through the following three phases.

1. Empirical/analytical design approach where Computational Combustion Dynamics based multi-dimensional calculations (CCD) provide qualitative insight.
2. Hybrid modeling that matched data from several diffusion, rich-lean and lean premix combustors.
3. The anchored design methodology that has been applied to recently introduced propulsion and aero-derivative industrial gas turbine engines.

Advanced unstructured CCD codes along with clustered PC/work stations, combustion models (both RANS- and LE-based) are at the critical threshold of providing the designer the tools they need to analytically design the entire combustion system to meet the design requirements. The rig and engine testing will be required for the sole purpose of verification and system-level refinement.

Combustion CFD – A Key Driver to Reducing Development Cost and Time

M S Anand* and C H Priddin#

***Rolls-Royce Corporation, Indianapolis, IN 46254, USA**
#Rolls-Royce plc, Derby, DE24 8BJ, United Kingdom

ABSTRACT

This paper will show how CFD has become embedded in the combustion design process, and how it has produced very significant process cost savings. We will show the way current models have been used, and will identify the most significant developments for further benefits to be obtained. Some of these involve code functionality; some involve changes to the development process itself.

As in all other industries, the big drivers today are reduction of time and cost to develop new products. These in turn demand that the number of hardware tests be reduced. This can only be achieved if it is easy to produce reliable, comprehensive results from analysis codes. Parameters of interest are pressure loss, temperature distribution, emissions, stability and life. If all these can be reliably predicted the role of rig testing is reduced to a single verification test on each new project. This ideal is not yet achieved – however considerable progress has been made recently and it is timely to reassess progress towards this goal.

Details will be shown which describe the choices of submodels appropriate for each parameter. There is also a balance to be drawn between minimising user input and minimising run time. Unstructured methods can produce meshes with little user input, but the resulting meshes are often very large and inherently slow to run, and the detail is provided in areas which may be of little interest. Structured codes on the other hand tend to be faster (often significantly so for the same mesh size) and more accurate on a similar grid, but require more user input to create. The paper will show how we have combined minimal user input with maximum grid quality.

The enhancements to the analysis process also bring the possibility to optimise the entire design cycle. For the analysis to be effective requires the creation of CAD solids early in the process. Initially these will contain little detail and will make full use of parametric modelling so that changes can be performed quickly, with the resulting effects on the flow being modelled directly with little user input. As the design develops, more detail will be added to these models to allow the design to be progressed. Since there will always be a solid model available this can also be used early in the manufacture planning process to start ordering fixtures and to optimise the manufacture process. Stress analysis is also possible, using geometry derived from the same model.

In the full paper several different examples will be shown of how engine projects have begun to benefit from this approach. These will include civil and military aero combustors, and a power generation combustor, all designed using results from CFD analysis. Examples of CFD application are presented in Figures 1 and 2.

All recent projects have benefited from these methods, often in more ways than expected. In one case the cost savings were such that the project would not have gone ahead without the reduced cost possible. The first rig test of a new combustor met all its design targets (except for those parameters not covered by the CFD). A cost-saving manufacture mod was introduced which could not have been assessed in time previously, saving a significant amount over the project lifetime. Potential failure mechanisms have been examined and proven safe.

Finally the outstanding issues will be addressed, with a view of the likely future developments in this field.

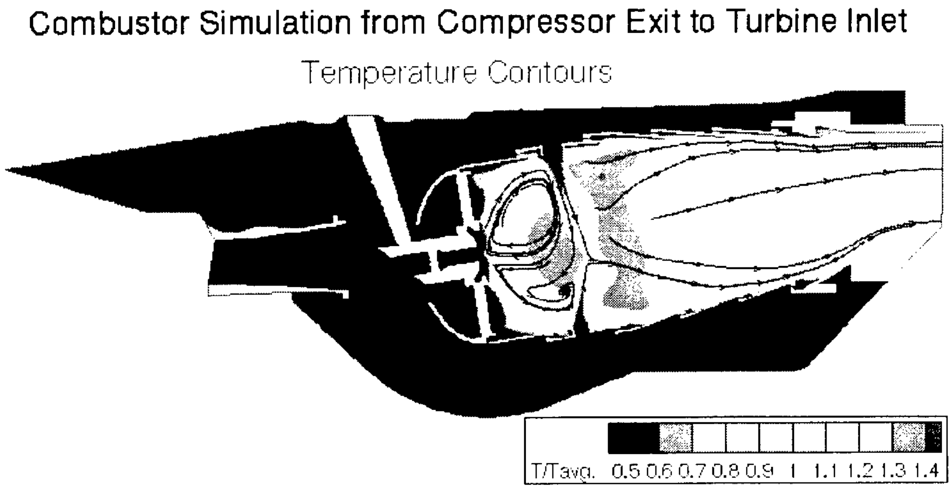


Figure 1: CFD simulation of a production combustion system from compressor exit to turbine inlet using Rolls-Royce 3-D analysis tool PRECISE. Such simulations provide valuable insight and guidance for combustion system design.

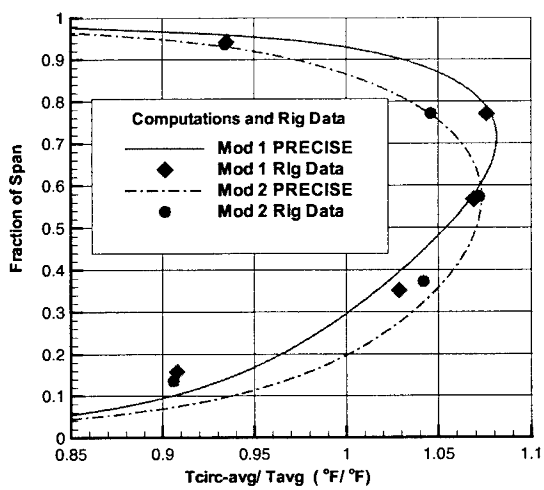


Figure 2: Computations with PRECISE match well with rig data for exit temperature for production combustors

ISABE-2001-1088

Integration of CFD and Low-order Models for Combustion Oscillations in Aeroengines

M. Zhu   A.P. Dowling and K.N.C. Bray

Department of Engineering, University of Cambridge
Trumpington Street, Cambridge CB2 1PZ, United Kingdom

Abstract

The pressure oscillations within combustion chambers of aeroengines are a major technical challenge to the development of high performance and low emission propulsion systems. The oscillations are driven by the resonant interaction between acoustic waves and unsteady combustion. The acoustic waves vary the air and fuel supplies and make the combustion unsteady. In return, unsteady combustion generates yet stronger waves. The resulting pressure waves can be so intense that they cause unbearable noise, vibration and even structural damage.

In this paper, an approach integrating a one-dimensional linear stability analysis and computational fluid dynamics (CFD) is developed to predict the modes of oscillation in a combustor and their frequencies and growth rates. Results from the CFD calculation provide the flame transfer function to describe unsteady heat release rate. Departure from ideal one dimensional flows are described by shape factors and we describe a procedure through which they can be determined. Combined with this information, low-order models can work out the possible oscillation modes and their initial growth rates. This work is a further development of our systematic investigation into the 'rumble' phenomenon, which occurs in aeroengine combustors at idle and sub-idle conditions and is typically in the range 50-120 Hz. However, the approach developed here can be used in more general situations for the analysis of any combustion oscillations.

ISABE-2001-1089

COMBUSTOR GEOMETRY OPTIMIZATION STUDIES USING CFD TECHNIQUE

By

G. Sivaramakrishna*, N. Muthuveerappan *, Venkataraman Shankar, T.K.Sampathkumaran*****
CFD (COMBUSTION & AFTER BURNER COMBUSTION) GROUP
Gas Turbine Research Establishment, C.V. Raman Nagar, Bangalore – 560093
INDIA

Abstract

The Computational Fluid Dynamics (CFD) plays a significant role with the magnificent power of super computers including parallel processors, CFD algorithms and codes. In fact, this is the single most advanced area in the recent decades currently enabling full Navier-Stokes computations on configurations such as combustor, which have been practically impossible a few years ago. Since the combustor full scale test facilities to simulate high pressure and high temperature flows is expensive and time consuming, the CFD has emerged for design optimization studies as it is based on comparatively less expensive and less time consuming numerical experiments.

The combustor geometry is given in Fig. 1. A three dimensional computational Body Fitted Co-ordinate(BFC) grid has been generated for a 20 degree sector of the combustor. This is because the combustor geometry on hand has 18 burners and one sector is used for the three-dimensional flow field computation. The computational grid, which is of structured in nature, has been generated using PHOENICS (Parabolic, Hyperbolic or Elliptic Numerical Integration code series) [1]. The grid has 153 cells in the axial direction, 105 cells in the radial direction and 10 cells in the circumferential direction amounting to a total number of cells of the order of 1,60,650. Fig.2 shows the grid generated for the given combustor geometry. Various blockages and liner openings are represented on actual area basis within a band of ± 5%. The flow in combustor is of elliptic in nature with appropriate boundary conditions. The continuity, momentum and energy equations are solved using answer code [2] and a four-step reaction mechanism has been employed for modeling combustion. The preliminary flow field analysis [3] of the given combustor geometry had indicated three main areas of concern, namely, peaky radial pattern factor exceeding the design intent by 0.08, high circumferential pattern factor exceeding the design intent by 0.18, primary zone flame discontinuity.

Before using the CFD technique for optimizing the combustor geometry, the analysis has been carried out for 4 sets of initial conditions for which full-scale combustor test rig, experimental results have been available. These have been presented in [4] and there has been found a good experimental and analytical match within an error band of 3 to 4%.

Having gained the confidence in using the "ANSWER" code, computational studies have been carried out for combustor geometry optimization. In this paper, attempts have been made to eliminate the areas of concern, mainly, peaky radial profile. The dilution hole configuration is altered and the temperature pattern at the combustor exit is studied. In some configuration, the pattern factor values have even worsened.

It could be observed from Fig.3 that the peaky radial pattern factor comes down to the tune of 0.15 and is thus giving desirable effect. Also the inner annulus mass flow rate has been jacked up to the tune of 2%. Thus, in this paper use of CFD has been illustrated for computation of the flow field in a given combustor geometry and optimizing the combustor geometry based on the CFD design changes and numerical experiments.

Reference

1. PHOENICS :

   Reference Manual, Part A & B, Concentration Heat And Momentum Limited, U.K., 1991.

2. ANSWER:

   Software Tool for Navier-Stokes, Energy and Mass Transport in Turbulent Reacting Flows, user's Manual, version 4.00 Rev. 2, Analytic and Computational Research Inc.1999

3. ACRI :

   Preliminary 3D Flow Field Analysis, ACRI/GTRE/957/02, March'2000

4. G. Sivaramakrishna, N. Muthuveerappan, Venkataraman Shankar and T.K Sampathkumaran:

CFD modeling of the Aero Gas Turbine Combustor, Paper presented at ASME Turbo Expo 2001, New Orleans, Louisiana, USA, 4-7 June 2001, ASME-2001-GT-0063.

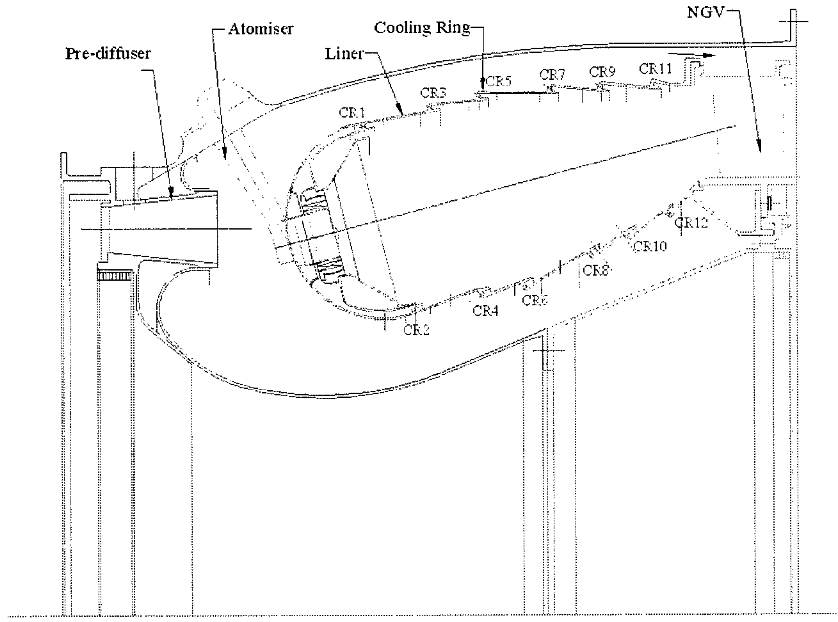


Fig. 1 Typical Combustor Geometry

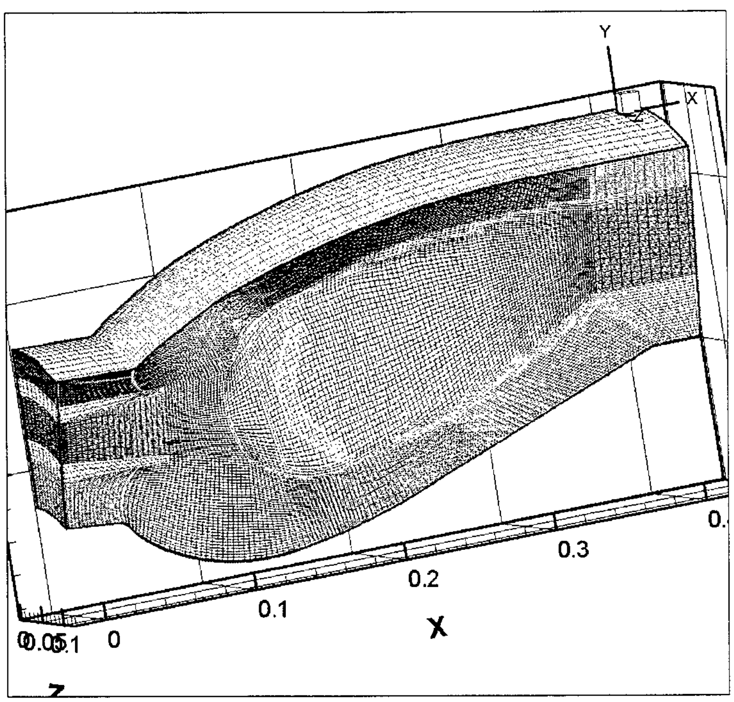


Fig. 2 Three Dimensional Combustor Grid

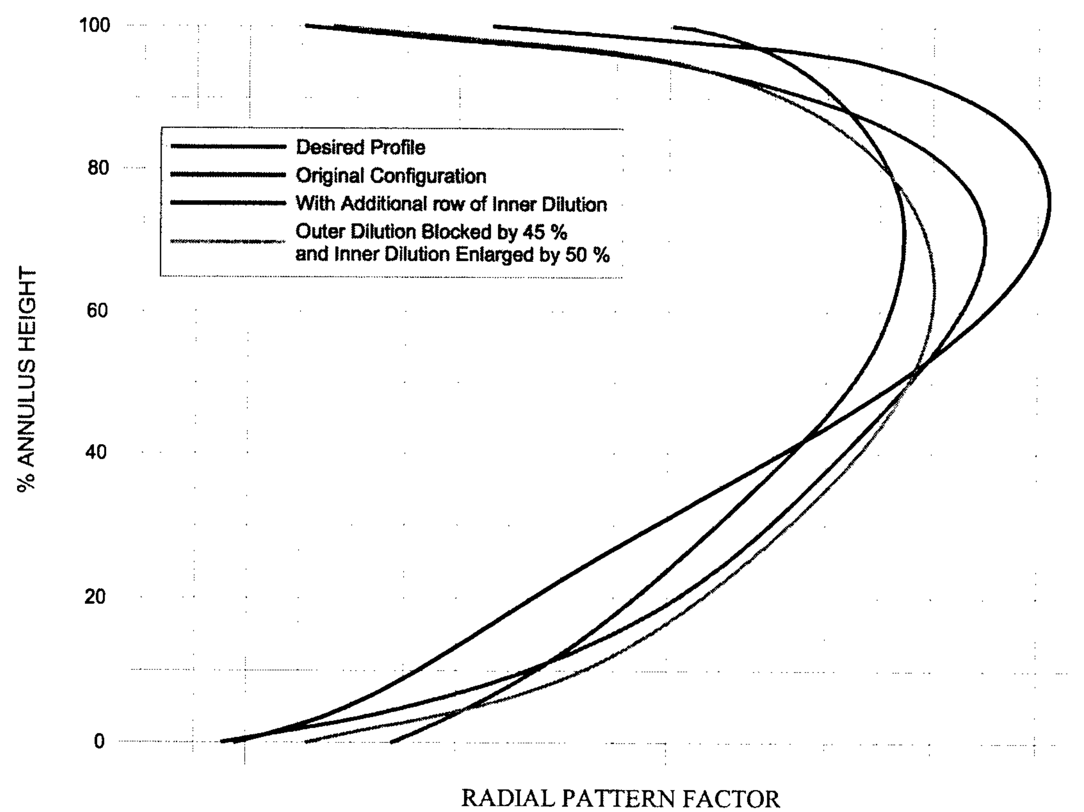


Fig. 3 Pattern Factor Improvement by design optimization using CFD Technique

ISABE-2001-1093

The Transition Zone in the Adverse Pressure Gradient Flow: Results from Stochastic Simulations

N Vinod & Rama Govindarajan
Fluid Dynamics Unit, Jawaharlal Nehru Centre for Advanced Scientific Research, Jakkur, Bangalore-560 064, INDIA.

E-mail: nvinod@jncasr.ac.in, rama@jncasr.ac.in

Abstract

We conduct stochastic simulations of spot generation and propagation in transitional boundary layers. We employ the hypothesis of concentrated breakdown as well as observations that spot growth is self-similar. When the breakdown of spots is prescribed to be random in time and the spanwise coordinate, the simulated results agree well with measurements of intermittency distributions in flat plate flow. In support of the scenario of regular breakdown proposed by Walker & Gostelow, it is shown that intermittency measurements in highly adverse pressure gradients are matched better by the prescription of a regular breakdown of spots.

ISABE-2001-1094

GLOBAL INSTABILITIES IN TRAILING-EDGE LAMINAR SEPARATED FLOW ON A NACA 0012 AEROFOIL

Vassilios Theofilis
DLR Institute of Fluid Mechanics
Bunsenstraße 10,
D-37073 Göttingen
GERMANY

and

Spencer J. Sherwin
Department of Aeronautics
Imperial College of Science, Technology and Medicine
Prince Consort Rd., South Kensington,
London, SW7 2BY
ENGLAND

Abstract

Steady laminar two-dimensional separated flow in the vicinity of the trailing edge of a NACA 0012 aerofoil has been analysed with respect to its global linear instability. Highly accurate base flows were obtained by state-of-the-art spectral/hp element numerical techniques. The partial-derivative eigenvalue problem governing amplification of non-Tollmien-Schliching small-amplitude three-dimensional instabilities, which are inhomogeneous in two and periodic in one spatial direction, was solved numerically. At the moderate Reynolds number monitored no unstable eigenmodes have been found. However, the spatial structure of the least damped global eigenmode is strongly reminiscent of its well-documented analogue in Howarth´s closed recirculation bubble flow[9]. This points to the existence of another, little explored, route to instability and laminar-turbulent transition that needs to be understood, if effective control of flow over aerofoils and turbine blades is to be achieved.

ISABE-2001-1096

Separation control in ultra-high lift aerofoils by unsteadiness and surface roughness

O.N.Ramesh[1+], H.P. Hodson[2] and N.W.Harvey[3]

[1] Department of Aerospace Engineering
Indian Institute of Science, Bangalore, India
[2]Whittle Laboratory, Cambridge University, U.K
[3] Turbine Engineering, Rolls Royce plc., Derby, U.K

[+] Contact Author. E-mail: onr@aero.iisc.ernet.in

Abstract
The high-lift philosophy (Howell at al. 2000) exploits the unsteady wakes which are shed from upstream blade rows to impact on the turbine blade under consideration and initiate laminar-turbulent transition of the suction surface fluid, thereby resulting in a total pressure loss reduction. This enables far higher levels of loading with unsteady flow than was previously possible in the traditional LP turbine bladings with a steady inflow. However, the efficacy of unsteady effects alone to achieve loss reduction seems to reach a saturation as the blade loading tends to ultra-high values. This is because such ultra high lift profiles have extremely strong deceleration zones on the aft part of the suction surface and the ill effects of the resulting large separation bubble (and sometimes even open separation) could be only partially contained by the unsteady effects. Hence the aim of this study is to explore if further loss reduction is possible for such ultra-high lift profiles by supplementing the unsteady effects by selective roughening of the aerofoil surface. It is shown in this experimental study that the combination of roughness and unsteadiness can indeed bring about a substantial reduction in total pressure loss. Measurements of surface static pressure distribution and wake traverse of total pressure are shown here which clearly show the substantial loss reduction due to the selective roughness patch in conjunction with the unsteady wakes. Hotfilm anemometry measurements of wall shear stress are analysed to arrive at a mechanism of loss reduction.

Keywords: *Ultra-high lift aerofoil, wake, roughness, unsteadiness, transition, turbulent spot, separation, loss reduction*

ISABE-2001-1097

EXPERIMENTAL INVESTIGATION OF AERODYNAMIC EFFECTS OF FILM COOLING ON A MODERN 3- DIMENSIONAL NOZZLE GUIDE VANE

Wies S.-H.[1]
Phon: +46 8 790 61 40
Email: sven@egi.kth.se

Fransson, T. H.
Phone: +46 8 790 7467
E-mail: fransson@egi.kth.se

Fax +46 8 204161
Chair of Heat and Power Technology
Royal Institute of Technology
Brinellvägen. 60
100 44 Stockholm, Sweden

Extended Abstract

Due to the necessity of more detailed three-dimensional data on the behavior of film cooled blades an annular sector cascade turbine test facility has been taken into service. The annular sector cascade facility is a relative cost efficient solution compared to a full annular facility to investigate three-dimensional effects on a non cooled and cooled turbine blade.

The aerodynamic investigations on the annular sector cascade facility are part of a broad perspective where experimental data from a hot annular sector cascade facility and the cold annular sector facility are used to verify, calibrate and understand the physics for both internal and external calculation methods for flow and heat transfer prediction.

Experimental investigations in terms of surface flow visualization and 5 hole pressure probe traverse of the influence of film cooling on the secondary flow effects and the looses of the cascade have been performed on a modern three dimensional nozzle guide vane with shower head cooling at the leading edge, four film cooling rows at the suction side, two film cooling rows at the pressure side and trailing edge ejection. The results of the flow visualization and pressure probe traverse show that the secondary flow region is only slightly effected by the ejection of low momentum (I = 0.36) cooling air. The cooling jets are deflected towards the hub, due to the low energy contents. With increasing mass flux ratio Y, respectively momentum flux ratio I the expanded secondary flow area at the trailing edge decreases. A rapid increase of the mixing loss at the midsection for ejection of high mass flow ratios in a highly accelerated flow at the suction side is observed. The coolant is seen, in every case, to increase the loss compared with the uncooled case. This is in accordance with the findings of most authors with regard to airfoil surface cooling, but the decrease in loss that is sometimes observed in the presence of trailing edge ejection is not observed here.

Some sample results are shown in Figures 1-3. In Fig. 1 the p_{02}/p_{01} distribution at the shroud for ejecting cooling air at the rows LE0-5, SS1-4, PS1-2 and TE is shown. For a mass flux ratio Y = 4.4% the total pressure loss strongly increase, but no material with a low energy content is transported from the suction side towards the flow passage. With increasing mass flow ratios of Y= 6.6 and Y=8.6 the flow field at the shroud is dominated by the shower head cooling air ejection. The flow field at the hub is shown in Fig. 2. By increasing the mass flow ratio Y the loss maximum shifts further towards the hub endwall and the wake gets thinner.

Fig. 3 shows the radial distribution of the kinetic energy loss coefficient. A strong increase of the mixing losses at the mid section can be observe with increasing mass flux ratio Y. The ejection of cooling air at the suction side into a high accelerated flow cause high losses.

[1] Current address: MTU Aero Engines
Heat Transfer/Turbine Cooling
München, Germany

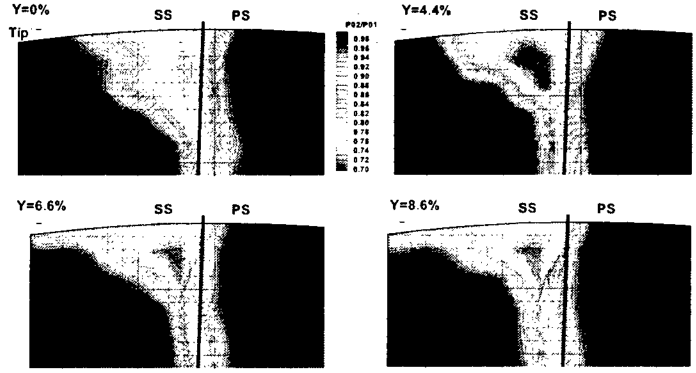


Fig. 1 p_{tot2}/p_{tot1} *distribution with cooling air ejection at row LE0-5, SS1-4, PS1-2 and TE*

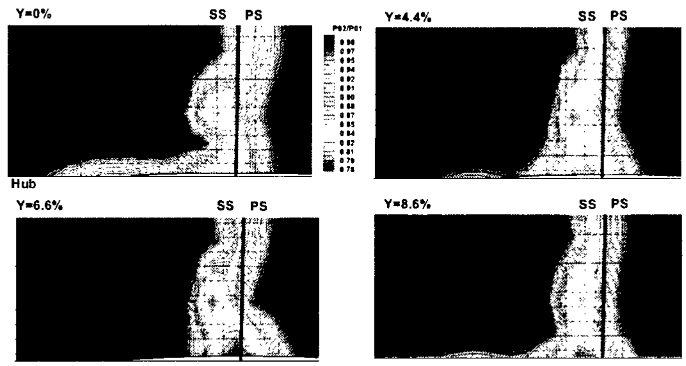


Fig. 2 p_{tot2}/p_{tot1} *distribution with cooling air ejection at row LE0-5, SS1-4, PS1-2 and TE*

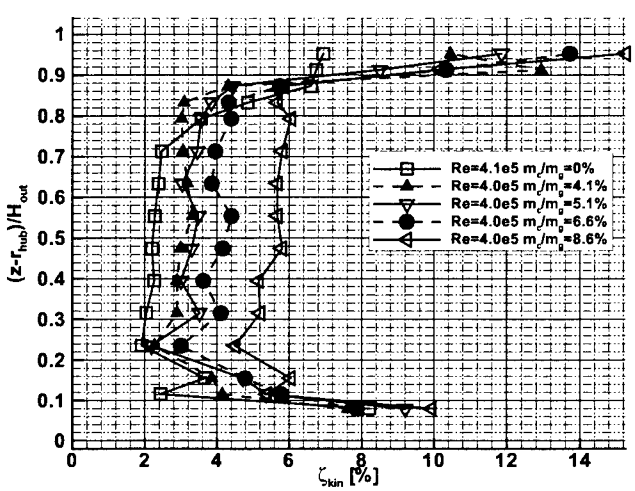


Fig. 3 Pitch wise mass averaged ζ_{kin} *distribution*

Keywords: turbomachinery, annular sector cascade, film cooling, secondary flow, losses

ISABE-2001-1099

Efficient Cooling Using Micro-channels

By

M.-N. SABRY
Mansoura University
Mechanical Engineering Department
Mansoura, Egypt

Extended Abstract

Channels of hydraulic diameter ranging from few to several hundred microns, called micro-channels, in which a highly purified fluid is forced, were relatively recently proposed as an efficient technique to convey large amounts of heat (up to 700 W/cm^2) from a small space. The application of this new technique is steadily growing in many modern fields including electronic, optical and medical devices. It can be extended to other areas like turbine blades.

Despite of a burst of publications in this area in the last decade, modeling of liquid flow and heat transfer in micro-channels is still an open problem. In fact, many investigators have experimentally proved that well established correlations used for normally sized channels do not apply in the case of micro-channels. Deviations increase as the channel size decreases.

In this work, observed experimental deviations will first be listed, followed by a critical review of different hypotheses advanced in the literature to explain them. A new hypothesis relying on surface roughness will be made, based on which simple models will be derived to predict most of the phenomena encountered in micro-channels.

In particular, the reduction of both friction and heat transfer compared to predictions of classical theory for low Re, as well as the increase of both Po and Nu numbers with Re in the laminar range, will all be explained. At relatively high Re, both Po and Nu may be higher than the classical theory as has also been observed. A criterion will also be derived for the transition to turbulence that predicts the observed early transition.

The model was successfully confronted with available test results from different authors giving both the order of magnitudes and the trends. It is important to note that the model contained no adjustable coefficients. It contained though two parameters having a physical meaning and thus an expected limited range of values. Very good agreement between model and experiments was obtained for parameter values lying in the middle of this range. Future work in this area may give a more precise value of these parameters depending on the nature of walls, liquid and machining procedure used.

ISABE-2001-1100

INVESTIGATION OF THE USE OF CFD FOR PREDICTION OF FREE JET FLOW AND SINGLE JET IMPINGEMENT HEAT TRANSFER

J M Brown[1], E D M Griffiths[2]
BAE SYSTEMS, Airbus UK
Bristol BS99 7AR
United Kingdom
S R Raghunathan[3], E Benard[4]
School of Aeronautical Engineering
The Queen's University of Belfast
United Kingdom

Abstract

A commercial CFD code is used to model both free jet flow and single jet impingement heat transfer. Relatively incompressible and compressible air jets are modelled. A standard k-ε turbulence model is employed as well as non-linear, quadratic and cubic, models and adjusted k-ε models. Predicted free jet flow velocities and temperatures are compared to measured values. Comparisons are also made between predicted and measured single jet impingement heat flux. The results show that standard k-ε and non-linear models overpredict the jet spread rate, however the radial spread of temperature is underpredicted. The sensitivity of heat transfer predictions to the initial jet turbulence intensity is also demonstrated. Predictions of jet flow field and heat transfer can be improved by adjustment of the turbulence model Cε1 value, however the optimised values are found to be different for the incompressible and compressible jets investigated.

ISABE-2001-1101

ADVANCES IN GAS TURBINE HEAT TRANSFER ANALYSIS USING CFD AND OPTIMISATION, IN CONJUNCTION WITH CONVENTIONAL FE METHODS

J. A. Dixon
CEng MRAeS
Rolls-Royce plc
Derby, England

J. A. Verdicchio
MSc (Oxon)
Rolls-Royce plc
Derby, England

A. B. Colley
AMIMechE
Rolls-Royce plc
Derby, England

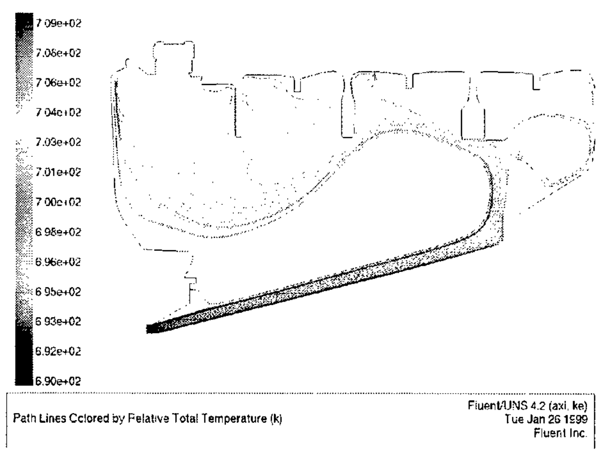
K.M. Tham
The University of Sussex
TFMRC
Brighton, England

Extended Abstract

Gas turbine manufacturers are constantly striving to improve their products to maximise customer satisfaction in terms of performance, reliability and cost of ownership, and in meeting regulation authority expectations for safety considerations and minimising environmental impact. Continuous improvement of the processes involved in engine and component design is a major element of the strategy to achieve these objectives.

Accurate temperature predictions are essential to the design optimisation of many gas turbine components. This paper provides examples of the application of recent developments in heat transfer boundary condition derivation and finite element model validation.

Computational Fluid Dynamics (CFD) is increasingly being used to determine cooling flow distributions and convective heat fluxes on a range of components. Validation of the CFD methodology for internal cavity heat transfer is also a key focus of major research programmes. In this paper results are presented for selected engine and test rig cavities. A fully coupled CFD/finite element thermal model solution is demonstrated.



The method has also been compared with the standard convective heat transfer correlations. Results for the heat loss of a disc rotating in free space are presented here since the free disc correlation is often used to determine heat transfer around compressor and turbine discs in engines.

A recent development in the application of optimisation techniques to the thermal model calibration process, has shown that significant savings in analysis time can be achieved for a given accuracy of 'match'. The optimisation process is described and sample results are presented from the calibration of a typical turbine disc assembly thermal model.

Finally the impact of these new analysis techniques on the derivation of thermal boundary conditions in gas turbine component cavities, and the implications for compliance with Airworthiness Authority regulations, is reviewed with respect to offering an improved temperature predictions validation strategy.

ISABE-2001-1102

Numerical Investigation of Heat Transfer and Film Cooling for Gas Turbine Applications

by

Martelli F., Adami P., Belardini E.
Energetics Department "S. Stecco" - University of Florence
Via S. Marta 3 – 50139 Florence, ITALY.

ABSTRACT

The CFD simulation of internal flows for turbomachinery applications usually ranges from the single blade investigation (the flow field, effects of blade twisting and bending, tip clearance, cooling injection etc...) to the representation of the whole stage and to the time dependent interaction of the rows in their reciprocating movement. With the enhanced complexity of the physical problem the accuracy allowed by the turbulent models become of primary importance.

Often the numerical solution of 3D turbulent unsteady flow requires computational resources not easily disponsable even in the modern single processor architectures, in spite of the enormous electronic and technological improvements in the second half of the century. Owing this limitations, the RANS modelling of the turbulent flow is still considered the best approach for industrial applications allowing at the same time a reasonable accurate and numerical affordable investigation of complex 3D flows. In this regard, despite their physical limitations, simple eddy viscosity closures based on the Boussinesq assumption are still widely used. Several attempts have been performed in the last two decades by researchers in order to compare the performances of the different turbulent closures available in litterature. Those investigations are generally performed using simple basic turbulent flows in the incompressible regime leading often to accurate predictions of the flow field characteristics. Anyway the rather different flow conditions encountered in the moder turbine blade rows impose the evaluation of the performances of models usually tuned for simpler flow configurations.

In the present work the application of different turbulent closures is considered for an hybrid flow solver consistent with unstructured and structured meshes. The main objective of the work is twofold. Firstly the capability and accuracy allowed by the unstructured approaches will be considered in order to simulate real turbine phenomena such as shock waves interactions, viscous boundary layers, heat transfer and coolant ejection/mixing effects. Then the comparison of four different turbulent closures will be undertaken for the investigation and prediction of the above mentioned phenomena for an high loaded NGV turbine blade and for investigation of advanced film cooling devices.

In order to compute practical flows at large Reynolds numbers, turbulence phenomena can be accounted through a classical RANS approach using different closure models: algebric Baldwin-Lomax, linear low Re *k-w* by Wilcox, non-linear *k-w* by Sofialidis and Prinos and low Re *k-e* by Launder and Sharma. Further different transition models are also applied and tested in conjunction with the previous closures in order to simulate the correct onset of turbulent transiction.

The unstructured code uses a finite volume discretization and a fully implicit time marcing approach. Numerical artificial dissipation is added in the finite difference scheme following the classical Jameson formulation while in the unstructured method a second order TVD upwind formulation is considered.

Key Words: CFD, heat transfer, turbulence modelling, film cooling

ISABE-2001-1103

Materials Development for Next Generation Aero Propulsion Turbine Engines

Robert E. Schafrik, Ph.D.
General Manager, Materials And Process Engineering Department
GE Aircraft Engines, Cincinnati, Ohio
robert.schafrik@ae.ge.com

Technology for aero-turbines has advanced considerably over the course of the last 50 years. Materials and processing technology have contributed significantly to this advancement. Partially in recognition of this, the US National Academy of Engineering identified High Performance Materials as one of the top 20 Greatest Engineering Achievements of the 20th Century. In the past, many of the material developments occurred because the materials community anticipated the needs of the turbine design engineers, who in turn were eager to exploit the advantages that new materials offered, even though the time to fully develop these materials oftentimes spanned a decade or more.

However, this paradigm is shifting 180 degrees. Future engine designs are certain to continue to evolve and change. As we consider what the aero-engine will "look like" in next 10-20 years, it is clear that some new materials technologies will be in the vanguard of enabling these changes. This paper will highlight four Grand Challenges for materials that must be addressed in order to meet future aero-engine needs. Each Grand Challenge has supporting enabling technologies that must be developed; examples of these are presented.

Taken together, these Grand Challenges and Enabling Technologies present the opportunities for the materials community to continue to provide major technology leadership to aircraft aero- propulsion. One hundred years from now, High Performance Materials will certainly be listed within the top 20 Greatest Engineering Achievements of the 21th Century!

ISABE-2001-1104

MATERIALS & PROCESSES FOR AFFORDABLE AND HIGH PERFORMANCE PROPULSION SYSTEMS

Dinesh K. Gupta
Pratt & Whitney
Materials & Processes Engineering
East Hartford, Connecticut 06108 USA

Abstract

Materials and processes have played a major role in enhancing the performance, improving the durability / maintainability, and reducing the cost of gas turbine engines over the past decades. These materials include: single crystal turbine airfoils, thermal barrier coatings, high temperature and high strength Ni shafts, damage tolerant and high creep strength powder metallurgy Ni alloy disks, hard-smooth compressor abradables, and others. Over this same period, turbine inlet temperatures have increased from 2000°F to ~ 3500°F, and engines can operate for much longer time intervals between overhaul. Military engines have achieved a thrust to weight ratio of greater than seven to one. Despite these advancements, materials development cost remains high and development cycle time remains long, 5 to 10 years. The lengthy materials development cycle is now incompatible with the engine development cycle (currently 3 years and heading toward 2 years in the near future). A recent focus of development efforts is to identify process improvements that accelerate the development cycle.

This paper discusses materials and processes that have a direct impact on affordability and performance without compromising quality,

durability, and maintainability. More specifically, recent trends and requirements in materials development to increase turbine inlet temperature and rotor speeds and reduce the weight of engine components will be discussed. In addition, affordability issues will be addressed including: design to producibility, the application of new processes to reduce cycle time, and the deployment of modeling and simulation in materials and process development.

ISABE-2001-1106

Development of Advanced Engine Materials in NASA's Ultra Efficient Engine Technology Program

Ajay K. Misra
NASA Glenn Research Center
Cleveland, Ohio 44135, USA

Abstract

Advanced high temperature materials are being developed in NASA's Ultra Efficient engine Technology (UEET) program to reduce CO_2 and NO_X emissions for future aircraft engines. The materials being developed include high temperature disk alloy, next generation of single crystal nickel-base blade alloy, low conductivity ceramic thermal barrier coating, ceramic matrix composite for combustor liner and turbine vane applications, and ultra lightweight structures with high temperature alloys. This paper will discuss material challenges for reducing CO_2 and NO_X emissions in aircraft engines and review progress made toward meeting these challenges.

Key words: Engine materials, nickel-base alloy, ceramic matrix composite, coatings, nanotube

ISABE-2001-1107

BENEFITS ASSESSMENT OF INSERTION OF ADVANCED STRCUTURES TECHNOLOGIES INTO AIRBREATHING ENGINES

Galib H. Abumeri[1] and Surendra N. Singhal[2]
QSS Group Inc.
21000 Brookpark Road
Cleveland, OH 44135

And

Christos C. Chamis[3]
NASA Glenn Research Center
21000 Brookpark Road
Cleveland, OH 44135

Abstract

Two studies are presented in this paper. The first study summarizes the benefits and debits of using advanced composites instead of conventional metals in the fan, high pressure compressor, and high and low pressure turbines of a high speed engine. The benefits assessment is carried out using the Engine Structures Technology Benefits Estimator (EST/BEST) software system. The use of composites instead of conventional metals enhances the component's structural performance and reduces its weight. Furthermore, most aircraft mission parameters, and maintenance and direct/indirect operating costs are improved with the exception of NOx emission and cost associated with advanced materials. The second study describes a probabilistic evaluation to assess the effect of scatter in the bleed ratio of high pressure compressor on the return on investment (ROI), a key economic factor. The compressor bleed is used to cool the cabin and the turbine. Typical results obtained show that the scatter in the bleed ratio for turbine cooling dominates the scatter in return on investment.

ISABE-2001-1108

DURABILITY TESTING AND CHARACTERIZATION OF SiC/SiC CERAMIC MATRIX COMPOSITES FOR GAS TURBINE COMBUSTOR APPLICATIONS

Mark van Roode
Solar Turbines Incorporated
San Diego, California, USA

Karren More
Oak Ridge National Laboratory
Oak Ridge, Tennessee, USA

David Brewer
NASA Glenn Research Center
Cleveland, Ohio, USA

Gary Linsey
United Technologies Research Center
East Hartford, Connecticut, USA

ABSTRACT

Significant progress has recently been made in evaluating SiC/SiC ceramic matrix composites (CMCs) for application as structural materials for gas turbine combustors. Under the US Dept. of Energy (DOE) sponsored Ceramic Stationary Gas Turbine (CSGT) program SiC/SiC CMC liners have been inserted in annular combustors of gas turbines of Solar Turbines Incorporated (Solar) operating at industrial sites. The liners are instrumental in lowering NOx emissions by leaning out the flame in the primary zone of the combustor and lowering CO emissions through the "hot wall" effect.

The SiC/SiC CMC combustor liners have been tested for over 33,000 hours in Centaur 50S engines at the Bakersfield enhanced oil recovery site of Texaco Exploration and Production, and Malden Mills the textile factory in Lawrence, Massachusetts. The liners were supplied by Honeywell Advanced Composites, Inc., and B.F. Goodrich Aerospace, and were fabricated using the chemical vapor infiltration (CVI) and melt infiltration (MI) processes. Significant surface degradation was detected in early engine testing because of accelerated oxidation in the gas turbine environment. Recent testing has been conducted with SiC/SiC CMC liners that had an environmental barrier coating (EBC) developed and applied by United Technologies Corporation (UTC). Selected liners have been analyzed nondestructively and destructively and corrosion degradation patterns have been established.

A high-temperature, high-pressure, mixed-gas furnace ("Keiser Rig") at Oak Ridge National Laboratory (ORNL) has been used to evaluate the long-term stability of different monolithic ceramics and CMCs under simulated combustor (high water vapor pressure) conditions. These materials have been tested with and without protective environmental barrier coatings (EBCs). All of the exposures to date have been conducted at 10 atm total pressure (1.5 atm H_2O) and ~1200°C for incremental 500 hour runs. Microstructural evaluation of the samples was conducted after each 500 hour exposure to characterize the extent of surface damage, to calculate surface recession rates, and to determine degradation mechanisms for each of the different materials. While the exposure time for most uncoated CMC samples has typically been 1000-2500 hours, some CMCs with EBCs have been exposed for as long as 6500 hours. The validity of this exposure rig for simulating real combustor environments was determined by comparing materials exposed in the test rig and combustor liner materials exposed for similar times in an actual combustor.

The NASA High Speed Research (HSR)/Enabling Propulsion Materials (EPM) program designed, fabricated and tested SiC/SiC slurry cast, melt infiltrated combustor liners in a sector rig for 200 hours, under High Speed Civil Transport (HSCT) engine operating conditions. The results of pre-test inspections, in situ evaluations and post-test macro and micro-structural evaluations will be discussed. Design details of CMC attachments will be reviewed.

The paper will describe the testing of the CMC liners in rigs and engines and the characterization of the CMC material during and after engine testing.

ISABE-2001-1109

POTENTIAL APPLICATIONS FOR SMART TECHNOLOGIES WITHIN GAS TURBINES

John R Webster
Strategic Research Centre
Rolls-Royce Plc.
PO Box 31, Derby
England DE24 8BJ

Abstract

SMART technologies are set to make a major impact on all aspects of engineering, consumer goods and the service industries. They will enhance the adaptability, performance and co-ordinated optimisation of many components and systems. This paper gives a brief overview of the developing technologies and applications. This is followed by an example of where a simple mechanical system using Shape Memory Alloys is being developed to provide a system which will adaptively respond to its environment to give noise improvements. Whilst there is still a need for technology development, the challenge is also moving on towards cost effectiveness, production techniques, servicing and certification issues.

ISABE-2001-1110

APPLICATION OF AUTOASSOCIATIVE NEURAL NETWORK ON GASPATH SENSOR DATA VALIDATION

Pong-Jeu Lu and Tzu-Cheng Hsu
Institute of Aeronautics and Astronautics
National Cheng Kung University
Tainan 70101, Taiwan
Republic of China

Abstract

Gaspath analysis holds a central position in the engine condition monitoring and fault diagnostics technique. The success of gaspath analysis, as concluded from previous investigations, depends mainly on the quality of the measurements obtained. No matter with classical Kalman filter method or contemporary artificial neural network approach, a high success rate of diagnosis can only be guaranteed when a correct set of measurement deltas is available. The objective of the present work is to design a genetic auto-associative neural network algorithm that can perform off-line sensor data validation simultaneously for noise-filtering and bias detection and correction. Neural network-based sensor validation procedure usually suffers from the slow convergence in network training. In addition, the trained network often fails to provide an accurate accommodation when bias error is detected. To remedy these network training and bias accommodation problems, a two-step approach is proposed. The first step is the construction of a noise-filtering and a self-mapping auto-associative neural network based on the backpropagation algorithm. The second step uses an optimization procedure built on top of these noise-filtering and self-mapping nets to perform bias detection and correction. Non-gradient genetic algorithm search is employed as the optimization method. It is shown in the present work that effective sensor data validation can be achieved for noise-filtering, bias correction, and missing sensor data replacement incurred in the gaspath analysis. This newly developed algorithm can also serve as an intelligent trend detector. True performance delta and trend change can be identified with no delay to assure a timely and high-quality engine fault diagnosis.

Keywords: Sensor Validation, Engine Condition Monitoring, Artificial Neural Network, Genetic Algorithm

ISABE-2001-1111

MODEL IDENTIFICATION-BASED FAULT ANALYSIS METHOD APPLIED T0 JET ENGINES

By

Da-Guang Chen and Zhi-Li Zhu
Beijing University of Aeronautics and Astronautics
Beijing, P.R. China

Abstract

The capability of Model Identification-Based engine performance diagnostics is evaluated. In general, the commonly used fault diagnostic methods such as Kalman Filter and Artificial Neural Networks are based on linear relationships between component performance indices and monitoring parameters. Though artificial Neural Networks can be trained by actual fault cases without using the linear relationships generated from the engine performance simulation code, to collect sufficient actual fault cases to cover all of the possible faults may not be practical. For some fault cases at some engine operating points, the non-linearity of the performance model is significant, and it can affect the accuracy of diagnosis. Moreover, the commonly used methods are good for single fault diagnosis and not very efficient for multiple faults, especially for quantitative fault isolation. The Model Identification-Based method presented in this paper is based on non-linear engine performance simulation code. This method applied to a twin-spool, mixed flow turbofan engine was investigated. For testing, fault cases were generated from the engine performance simulation code, and the monitoring parameters so generated were contaminated by noise using a noise model to make the testing more realistic. The investigation shows that this method could isolate not only single fault but also multiple faults quantitatively with high accuracy. This method can also be applied to engine performance modeling based on experimental data.

ISABE-2001-1112

A FUZZY LOGIC INTELLIGENT SYSTEM FOR GAS TURBINE MODULE AND SYSTEM FAULT ISOLATION

By

Ranjan Ganguli
Department of Aerospace Engineering
Indian Institute of Science
Bangalore-560012, India

Abstract

A fuzzy logic intelligent system is developed for gas turbine fault isolation. The gas path measurements used for fault isolation are exhaust gas temperature, low and high rotor speed and fuel flow. The fuzzy logic system uses rules developed from a model of performance influence coefficients to isolate engine faults while accounting for uncertainty in gas path measurements. Tests with simulated data show that the fuzzy system isolates faults with an accuracy of over 90 percent with only the four cockpit measurements. However, if additional pressure and temperature probes between the compressors and before the burner that are often found in newer jet engines are considered, the fault isolation accuracy rises to as high as 98 percent. In addition, the additional sensors increase the fault isolation success rate as quality of the measured data deteriorates.

Keywords: fuzzy logic, fault isolation, gas turbine, intelligent system

ISABE-2001-1113

Smart TBC's for Gas Turbines

By

J P Feist and A L Heyes
Department of Mechanical Engineering,
Imperial College of Science Technology and Medicine,
London, UK

K. L. Choy and J. Mei
Department of Materials,
Imperial College of Science, Technology and Medicine,
London, UK

Abstract

The research described in the paper is concerned with the development of smart materials for use as thermal barrier coatings (TBC) in high temperature regions of gas turbines. These materials will incorporate the thermal insulating properties of conventional TBC's but will exhibit additional instrument properties enabling various attributes of the coating, including temperature, heat flux and erosion, to be sensed optically by interrogating the luminescent emission from the coating when illuminated with UV light. Such coatings would be of great utility in the design and development of gas turbine components and of cooling schemes associated with them and could provide on line feedback of conditions within the engine or information pertinent to component lifing by interrogation during routine maintenance interventions. In order to give TBC's instrument properties the ceramic material from which they are made, typically yttria stabilised zirconia, is doped with a small amount of a rare earth element such as europium (Eu). This effectively converts the material into a thermographic phosphor. These materials have been used for many years to enable surface temperatures to be measured over a wide range. Development of the technique, by the authors, for application in high temperature regions of gas turbines is also underway and is discussed in a separate paper.

Thermographic phosphors consist of a ceramic host matrix with a lanthanide ion dopant. When illuminated with UV light they exhibit luminescence which is temperature dependent by virtue of variations in the relative intensities of distinct emission lines or in the time constant of the exponential decay which occurs once excitation has ceased. YAG:Dy, YAG:Tb and Y_2O_3:Eu are examples and have been selected by the authors as phosphors suitable for use in high temperature gas turbine applications since each has a dynamic range extending to in excess of 1000°C. In each case the active element resulting in temperature sensitivity is the rare earth dopant ie Dy, Tb, Eu. The authors have investigated the possibility of producing a smart TBC material by producing yttria stabilised zirconia doped with europium (YSZ:Eu) in powdered form. This material was investigated and shown to be temperature sensitive with a dynamic range extending to 800°C (Feist and Heyes (2000)).

In the current paper we will describe the production of prototype smart TBC coatings using a new and novel coating technique developed in the Materials Department at Imperial College. The coatings produced consist of a thin layer of YSZ:Eu and a similar layer overcoated with a 50μm layer of un-doped YSZ. These coatings have been interrogated and shown to exhibit temperature sensitivity similar to powdered YSZ:Eu and it has also been shown that the temperature of a sub surface layer can be determined. The concept of a smart TBC has therefore been demonstrated and future work is now required to optimise the material composition and coating parameters to develop the technology to a production standard.

References

Feist J.P, Heyes A.L. Europium doped yttria stabilized zirconia for high temperature phosphor thermometry. Proc. Instn. Mech. Engrs. Vol. 214 Part L pp.7-12.

ISABE-2001-1114

Multistage Parallel Numerical Analysis for High Pressure Ratio Compressor with A Novel Computational Domain Extension Approach

Zhao Xiaolu

Institute of Engineering Thermophysics, Chinese Academy of Sciences

P.O.Box 2706,Beijing 100080, P.R.China

Extended Abstract

This paper described the multistage parallel Navier-Stokes solver based on the methodology of deterministic stress proposed by Adamzeck[1]. The deterministic stress is used to ensure continuos physical properties across the interface planes and to account for the average wake blockage and mixing effects due to the upstream blade rows. The coupling between the neighboring rows is implemented by means of a novel computational domain extension approach along with an approximate model to deal with deterministic stress distribution in axial direction.

The novel extension method was proposed as follows: The upstream calculation domain of the downstream blade was extended from the interface plane about a half or one third axial chord, as well as for the downstream extension for the upstream blade. Both of the two extension zones constructs a flow channel, within which an inviscid throughflow calculation allows radial boundary condition profiles to be transferred between the inlet of downstream domain and the outlet of the upstream domain. This axisymmetric Euler code is very similar to the traditional axisymmetric approach. The difference between the two approaches is only that the former is implemented in a reverse direction of the fluid flow to connect the upstream and downstream blades calculation. In this case, there is no need to adjust the back pressure level at calculation domain outlet for each blade row.

This code is applied to a 7-stage high-pressure ratio compressor at the design point. The computation is performed on DAWNING 1000A parallel computing system. Each blade row is parallelized using a domain decomposition method and the interface boundary conditions are communicated between blade rows at every iteration step. A standard "H" net is used, and each blade channel has 99 points in axial direction, 27 in the span and blade-to-blade directions.

The calculated Mach number contours on the midpitch of the first 5-blade row are shown in Fig.4. The calculated results extended domain on extended domains are compared with the results obtained from the reverse axisymmetric approach. The symmetric property of the iso-Mach number distribution on both side shows that the extended domains are well connected in terms of the reverse axisymmetric solutions.

Calculated Mach number counters at 90% span of first rotor with different inlet computational domain extension are shown in Fig. 7. These results demonstrate that the extension of computational domain plays an important role in correctly capturing the shock wave systems.

ISABE-2001-1116

DECISION SUPPORT SYSTEMS FOR THE AERODYNAMIC PERFECTION OF TURBOMACHINERY ROWS

M. L. Ugryumov, A. Yu. Sokolov, V. V. Kalinin, A. M. Tsegelnik
National Aerospace University named after N. E. Zhukovsky
"Kharkov Aviation Institute",
KhAI, 17 Chkalov Street, Kharkov, 61070, Ukraine
Fax: +380 572 441 131; E-mail: mlu@xai.kharkov.ua

Abstract

Turbomachine is a complex technical system with great number of parameters, which affect on efficiency of it operation. There are a lot of efficiency criterions, which are often in contradictions with each other. Therefore eventual result of designing depends on design decisions made on the stage of a requirement specification development and formation of the turbomachine basic characteristics. The international experience shows that at these stage decision-makers along with formal mathematical models also utilizes expert evaluations, data about existing analogs and intuition.

This consideration and constantly growing requirements to quality and terms of design was our motivation for developing DSS for the aerodynamic perfection of turbomachines.

Presented system allow to accumulate the information on available analogs, their characteristics and design features in a database, to conduct the database analysis with purpose of rules discovery in data. The mathematical models supplemented by detected rules and expert knowledge are supplied throw the interface block in the knowledge base. This knowledge base is exploited by the inference mechanism of the fuzzy logic controller. A main task of the fuzzy logic controller is providing feed back for optimization process.

The theoretical analysis of spatial boundary layer formation features on surfaces, limiting a blade passage, allows us to suppose necessity of usage of known ways of effect on a boundary layer to prevent separation. In particular, computational methods developed by authors may be used for: the argued selection of position of direct effects on blade and endwall surfaces (generators of longitudinal vortices, slots, injection, extraction); the solutions of a variational boundary value problem of aerodynamics to optimize the shape of a blade passage by the solution of a number of direct problems.

In the test example three-dimensional analysis of a flow in a fan stator blade passage carried out, and also a number of measures for flow separation elimination and increase efficiency is adopted.

ISABE 2001-1117

Numerical optimization for turbomachinery blades aerodynamic design using a gradient method coupled with a Navier-Stokes solver

Stéphane BURGUBURU*, Clément TOUSSAINT*, Gilles LEROY**
***ONERA, Applied Aerodynamics Department**
BP 72 92 322 Châtillon, FRANCE

****TURBOMECA, Aerothermodynamic Department**
64511 Bordes Cedex, FRANCE

ABSTRACT

This paper describes a numerical optimization code for 2.5D and 3D turbomachinery blades using a constraint gradient procedure coupled with a Navier-Stokes solver. Mesh deformations are ensured by parametric geometry deformations (Bézier curves) for both approaches. Objective functions and constraints are defined with a momentum average of Navier-Stokes computation results. A 2,5D transonic compressor rotor blade optimization is presented with a significant isentropic efficiency improvement. A 3D static turbine blade optimization is then described. Important reduction of losses and hub flow separation is obtained by applying lean deformation on the blade geometry.

ISABE-2001-1118

Computations of supersonic flow through compressor cascades

A. T. Sriram & Joseph Mathew
Department of Aerospace Engineering,
Indian Institute of Science
Bangalore 560012, INDIA

K. S. Ravichandran
Computational & Theoretical Fluid Dynamics Division
National Aerospace Laboratories,
Bangalore 560017, INDIA

Abstract

We consider the numerical prediction of flow in compressor cascades developing from supersonic inflows. Solutions were obtained using a finite-volume MacCormack method and a Roe scheme with MUSCL pre-processing, coupled with the k-ω turbulence model. Two experiments conducted at Detroit Diesel Allison (DDA) and ONERA through cascades of the same ARL-SL-19 blades were simulated. Predictions of blade surface pressures and wake profiles are in very good agreement for the DDA test case. Roe's method gave cleaner solutions. Predictions of the ONERA test (off-design) were less reliable; differences in shock locations were small, but there was a sensitive response from the suction surface boundary layer.

ISABE-2001-1119

High Pressure Combustion Research at the Air Force Research Laboratory

Dale T. Shouse, Jeffrey S. Stutrud, and Charles W. Frayne
Air Force Research Laboratory
Propulsion Directorate
Wright-Patterson AFB Ohio 45433 USA

Abstract

This paper presents the details of a high-pressure combustion research facility located at Wright-Patterson Air Force Base. The facility was designed for the experimental exploration of advanced combustor concepts under conditions simulating actual gas turbine engine operation. This facility is considered to be a unique research tool that provides a wide variety of test information for evaluating candidate combustor concepts using a range of diagnostic tools. These include conventional thermocouples, pressure transducers, gas sample rakes, and high-speed movies.

The facility also features advanced laser diagnostics that provide non-intrusive probing of the combustor flow-field. The facility can accommodate a range of test articles from single-cup test rigs to multi-dome 60-degree combustor sectors

.

ISABE-2001-1122

Optimisation of Combustor Wall Heat Transfer and Pollutant Emissions for Preliminary Design Using Evolutionary Techniques

J.M. Rogero P. A. Rubini*
Department of Automotive, Combustion and Energy Engineering,
School of Engineering
Cranfield University, MK43 0AL, Bedfordshire, UK

Abstract

This paper presents the conception and use of a design optimisation toolbox based on evolutionary techniques, for the preliminary design of gas turbine combustors. The toolbox has been designed to be interfaced with existing analysis packages and to perform optimisation in parallel over a network. The combustor design optimisation capabilities of the toolbox are demonstrated by automating the achievement of twenty-five performance targets for a combustor design whilst performing minimisation of wall cooling flow and NOx emissions.

ISABE-2001-1126

DEVELOPMENT OF SMALL GAS TURBINE ENGINES- HAL'S APPROACH

by

R.C. Panda, P. Arunachalam, C. Rangaswamy & S. Rengaswamy
Engine and Test Bed Research & Design Centre, Hindustan Aeronautics Limited, Bangalore, India

ABSTRACT

Small gas turbine engines are used in a great variety of ways and their design does not vary much from designing engines with high thrust or shaft power. In practice, however, there are many limiting parameters like the tip clearance, boundary layer thickness, surface roughness etc. which complicate the design of small gas turbine engines. Engine & Test Bed Research & Design Centre (ETBRDC), Hindustan Aeronautics Limited (HAL), India with a nuclear task of research and development of small gas turbine engines started functioning in the 1960. This paper presents the small gas turbine engines successfully designed and developed by ETBRDC, their broad specification, technological highlights & the challenges the centre has to face before completion. The spin-offs of these projects and also the approach that HAL wish to follow in fulfilling the needs of self sufficiency, indigenisation and lower development cost are also brought out.

Key Words :
1. Small Gas Turbine
2. Gas Turbine Starters
3. Small Turbojet Engines

ISABE-2001-1128

PERFORMANCE OF MODERN STOVL FIGHTER POWERPLANTS.

Jose Luis Herrero, Dimitris Myzithras,
Pericles Pilidis, Roy A. Cookson

School of Engineering, Cranfield University
Cranfield, Bedford, U.K.

ABSTRACT

The aim of this paper is the description of the performance characteristics of powerplants intended for modern STOVL fighters. The intended application for the propulsion system investigated here is a combat ASTOVL aircraft. **of** Two powerplants are examined in this paper, as they appear to be the most viable solutions for a combination of high performance, increased efficiency and STOVL capability. These powerplants are the lift-nozzle and the low pressure shaft driven lift fan. Both powerplants involve the integration of the same core, with different approaches for providing vertical thrust.The main cycle features are estimated and a prediction is made of the performance of the engine in a wide range of operating conditions.

ISABE-2001-1131

Fault Detection & Isolation Studies During Qualification Testing of a High Pressure Turbine Disc

Giridhar R., Sanjay Barad and P. S. Kasthuri Rangan
Gas Turbine Research Establishment
Bangalore, India

The present paper deals with the Fault Detection and Isolation – FDI activity carried out on a high power shaft test rig experiencing high vibrations at certain operating speeds of rotor during qualification testing. This rig consists of a test rotor driven by a DC motor through a step-up gearbox. Being a supercritical operation of test rotor the risk involved is high. This demands for strategic approach in problem identification and solving.

The FDI activity discussed in the paper involves integrated studies in vibration signal analysis, Experimental Modal Analysis - EMA and analytical studies on estimation of cause for high vibration, in other words problem identification, input characterization and response measurement is undertaken.

A step by step investigation procedure to ascertain the cause of high vibration is laid down and the elimination process to establish root cause is discussed in detail. Various possible remedial measures are discussed for its effectiveness.

The results accrued from the various tests are well corroborated, which in turn demonstrates the authenticity of test procedure adopted. Remedial actions suggested are incorporated. Finally, the feed back results indicated and conclusions drawn with minimum of details.

ISABE-2001-1132

PROBABILISTIC CRACK INITIATION LIFE ESTIMATION OF A GAS TURBINE DISK

R.W. Shende, Associate Professor,
M.C. Mujumdar, Former Research Scholar,
N. Ambalkar, Former Graduate Student.

Aerospace Engineering Department, Indian Institute of Technology- Bombay, India 400 076.

Abstract

The design optimization objectives of critical components are served eminently by application of the probabilistic concepts for life estimation, as opposed to the deterministic methods, which involve imprecise data and approximations with the consequence of large safety margins. The reported exercise in life estimation deals with the concrete case of the turbine disk of the high pressure rotor of a twin-spool aircraft gas turbine engine. The procedure for calculation of the `crack initiation life' of the disk, together with reliability is based on the Weibull Weakest Link Theory, using experimental material fatigue data with inherent scatter.

ISABE-2001-1133

A Physics-Based Approach for the Detection of Cracks in Rotating Disks

Héctor M. Rodríguez and Darren Hallman
GE Corporate Research and Development Center
Niskayuna, New York 12309 USA

Mark Butz
GE Aircraft Engines
Cincinnati, Ohio 45215 USA

Extended Abstract

A physics-based approach for the detection of cracks in rotating disks is presented. The approach takes advantage of the distinct behavior in the vibration response of cracked rotating disks. In particular, radial-axial cracks induce a unique vibration response as they open due to tensile hoop stresses caused by centrifugal loading. The crack opening forces a redistribution of the disk mass. This mass redistribution yields an additional unbalance that is proportional to the square value of the speed, and hence the resulting crack-induced unbalance force is proportional to the fourth power of the speed. This unique unbalance response characteristic brings the opportunity to implement an on-line monitoring system to detect early stage disk cracks by measuring the vibration response of the rotating assembly at non-invasive locations (e.g., bearings). The present work investigates the mechanism of crack-induced response in rotating disks as part of the development of the crack detection system. The study includes both finite element modeling and the development of an analytical approximation of the crack-induced response in disk-shaft systems. The developed analytical formulation allows parametric comparisons without the need of time-consuming FE analyses. As shown in the analysis, in addition of its speed dependence, the crack-induced unbalance is proportional to the second power of the crack length, the disk diameter, and the disk mass density, respectively.

ISABE-2001-1134

PROGRESS OF THE JAPHAR COOPERATION BETWEEN ONERA AND DLR ON HYPERSONIC AIRBREATHING PROPULSION

By

Ph. Novelli[1], D. Scherrer[2], ONERA, Chatillon - France
W. Koschel[3], DLR Lampoldshausen - Germany

Extended Abstract

In 1997, French ONERA and German DLR started a common research project to pursue, beyond the national programs PREPHA and Sänger, the study of high speed airbreathing propulsion.

The project focuses on the dual mode ramjet, the goal being, on one hand, to design and test at ground such an engine, and , on the other hand, to study its propulsive balance on a vehicle.

Activities are organized around the blue-print of a possible experimental vehicle, that provides a solid context for the studies and a basis to synthesize the results through the "thrust minus drag" budget.

An experimental dual mode ramjet has been designed and tested in 2000, in ONERA ATD5 facility, to validate the engine concept and design process. To support engine study, basic research are carried out on supersonic combustion. They contribute to the validation and development of physical models and numerical tools.

The aerodynamic components of the propulsion system are studied, the goal being to define possible configurations for the inlet and the nozzle, but also to study the integrated operation of the whole propulsion system and, at last extend, to predict the vehicle performances.

All the research carried out in the frame of JAPHAR contribute to the acquisition of the required know-how for the development of an experimental vehicle to demonstrate DMR performances in flight, in the frame of a larger project.

The paper presents a synthesis of the latest progress of the project and gives the outstanding results obtained up to now.

[1] Long Term design and System Integration Department, Project Manager
[2] Fundamental and Applied Energetic Department
[3] Professor, Director of DLR Lampoldshausen

ISABE-2001-1135

Testing and weighing of the JAPHAR dual mode ramjet engine

O. Dessornes[1], D. Scherrer[2], P. Novelli

The scramjet is the main foreseen solution for the propulsion of airbreathing vehicles beyond Mach 6. Most of its potential applications require that the airbreathing propulsion phase could be used for a part of the initial acceleration of the vehicle. This explains why the dual mode ramjet that works successively in subsonic and then in supersonic combustion regimes has been chosen as a central axis of the JAPHAR program which was jointly engaged in 1997 by ONERA and DLR, after the PREPHA and Sänger program, to pursue researches on hypersonic airbreathing propulsion.

The studied concept is a dual mode ramjet, with fixed geometry, that has a staged combustion chamber in order to allow the different combustion regimes – subsonic with thermal throttling, transonic and then supersonic – to take place in the same geometry depending on the flight Mach number (fig. 1). The upstream engine part is slightly diverging and is adapted to the supersonic combustion regime, whereas the downstream part, which is much more divergent, is designed for the subsonic combustion regime or the transition between the two modes. This dual mode ramjet, which has been developed on the basis of the PREPHA work, is hydrogen fuelled and is conceived for engines of large dimensions which led to inject the fuel thanks to struts. In the frame of this project, the engine has been designed in order to propel an experimental vehicle the mission of which is to demonstrate in flight the propulsive balance of a dual mode ramjet. Integral 0D analysis trealised from the forebody to the nozzle has allowed to determine the vehicle engine design on the complete trajectory, from Mach 4 to 8.

Taking into account the capacities of the available facilities at ONERA for scramjet ground testing, an experimental engine with reduced dimensions has been extrapolated from the vehicle engine for the tests.

The inflow cross section of the engine is 100 mm x 100 mm while the vehicle chamber entrance section is 400 mm x 100 mm. Thus, the height of the chamber can be respected on the model but the injection system has been modified to be adapted to the reduced width. As a result, the first injection stage has only one strut and wall injectors whereas the flight chamber has multiple struts. The second injection level is composed by two struts. The engine is 2.4 meters long, which correspond to the length of the flying engine.

The operation of the experimental chamber has been studied by numerical simulations firstly thanks to 1D computations on then thanks to the MSD code which is developed by ONERA. Navier-Stokes computations have been performed on an engine mesh that is composed of a bit more than 700 000 meshes. They show that a fully supersonic combustion regime should be attained at Mach 7.6 for an equivalence ratio equal to 1 and 80% of the fuel injected Through the first injection level. On the contrary, at Mach 4.9, the flow becomes subsonic through a shock and a thermal throat takes place near the exit of the chamber. For this combustion regime, the computations have showed that the position of the recompression system associated to this regime could be driven by the injection repartition between the different injection levels. This possibility has also been studied during the tests.

A complete test campaign has been carried out in 2000 on this test chamber, in the ATD 5 test cell of ONERA, for flight Mach numbers between 4.9 and 7.6 (fig. 2). Associated 3D computations have shown good agreement between the predicted and the experimental pressure profiles has could be seen figure 3 for the Mach 7.6 case.

On this basis, a new test campaign will be carried out in the ATD 9 test cell to weigh the engine for the most promising injection configurations found in the previous tests. This test campaign will allow to determine the exit stream thrust with an expected accuracy of 5%. Furthermore, the principal flow exit characteristics such as the mean Mach number, the density, the speed or the stagnation pressure will also be obtained.

[1] ONERA – BP 72 – 29 Avenue de la Division Leclerc – 92322 Châtillon Cedex, France. Tel : 01 69 93 60 24. Fax : 01 69 93 61 62. E-mail : olivier.dessornes@onera.fr
[2] ONERA – BP 72 – 29 Avenue de la Division Leclerc – 92322 Châtillon Cedex, France. Tel : 01 46 73 43 21. Fax : 01 46 73 41 47. E-mail : dominique.scherrer@onera.fr
[3] ONERA – BP 72 – 29 Avenue de la Division Leclerc – 92322 Châtillon Cedex, France. Tel : 01 69 93 63 08. Fax : 01 69 93 63 00. E-mail : philippe.novelli@onera.fr

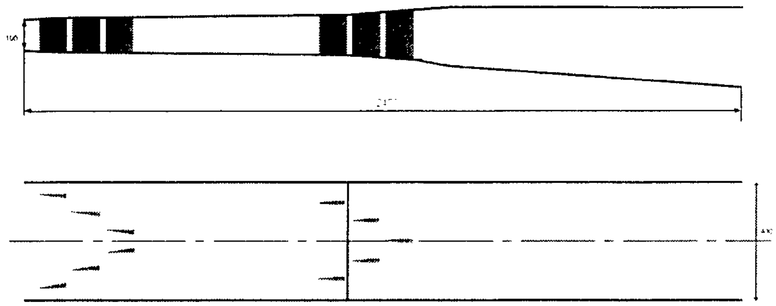


Figure 1 : scheme of the double combustion chamber DMR concept

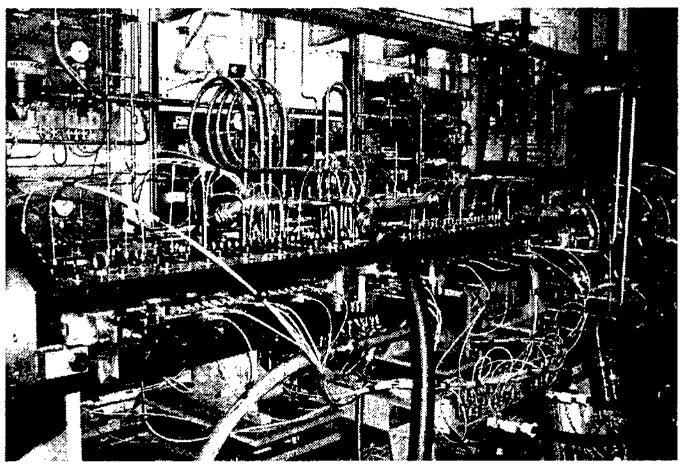


Figure 2 : the JAPHAR dual mode ramjet in the ATD5 facility

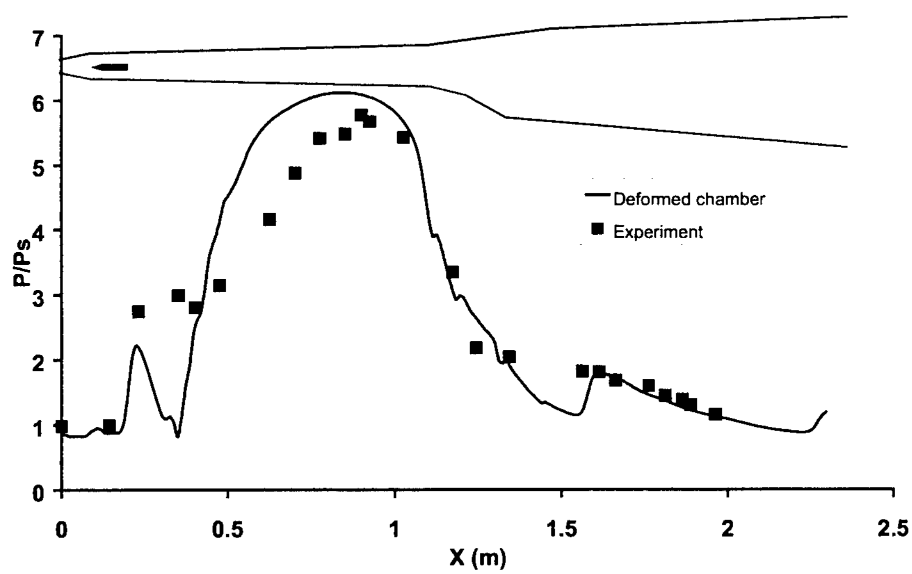


Figure 3 : Computed and experimental pressure profiles obtained at Mach 7.6

ISABE-2001-1136

INLET DESIGN FOR JAPHAR FLIGHT TEST VEHICLE

Ph. Duveau (ONERA Châtillon, France)
H. Esch, K. Triesch (DLR Cologne, Germany)

Abstract

In the frame of JAPHAR project different inlet concepts were designed in order to feed a dual mode ramjet engine propelling a flight test vehicle between Mach 4 and 8 and burning hydrogen as a fuel. The mixed compression inlets studied by DLR can be operated without bleeding and provide self starting capability at low Mach number, but, due to their high design Mach number (around 6) a large amount of flow spillage occurs at the lowest bound of the flight range. ONERA focused on internal compression inlets requiring bleeding and variable geometry to insure starting, but which could easily be operated with variable geometry to cover a wider Mach number range with view to future applications of airbreathing propulsion.
The paper gives the most significant experimental results obtained in H2K (DLR) and S3MA (ONERA) wind tunnels and put the emphasis on CFD validations performed by ONERA on the isolated internal compression inlet.

ISABE-2001-1137

Rationale for Fan-Supercharging the Hydrogen-Fueled Hypersonic Airbreathing/Rocket Combined-Cycle Engine

William J.D Escher, SAIC - Huntsville, Alabama, U.S.A.

ABSTRACT

For Spaceliner-class orbital transports powered by airbreathing/rocket combined-cycle propulsion (CCP) systems, integration of a turbofan-based "fan-supercharger" element is deemed essential. Beyond its ascent-flight performance benefits, the need for a powered descent and landing capability is met, as well as the provision of numerous important adjunct-function advantages. Especially for non-staged (SSTO) systems, the fan subsystem, as will the overall engine, will use cryogenic liquid hydrogen fuel. While adequate operating-hardware experience exists for developing the hydrogen-fueled turbofan subsystem, other fan-integration aspects remain at the conceptual design level. This prior work should now be expanded upon by the aero-engine and hypersonic propulsion development communities.

ISABE-2001-1138

COMPUTATIONAL STUDY OF FLOW IN A ROCKET BASED COMBINED CYCLE (RBCC) ENGINE INLET

D. R. Reddy
NASA Glenn Research Center
Cleveland, Ohio 44135 USA

and

D. Sree
Tuskege University
Tuskege, Alabama 36088 USA

Abstract

A two-dimensional axisymmetric viscous flow analysis of a rocket-based combined cycle (RBCC) engine inlet is performed using a time-marching numerical scheme to solve the Reynolds-averaged Navier Stokes equations. The flow configuration is a subscale model of the engine inlet which was previously tested in the 1X1 supersonic wind tunnel at NASA Glenn Research Center. The computed results are compared with the experimental data which include static pressure profiles along the centerbody and the cowl surface of the inlet. Comparison of the computed results with experimental data shows that the numerical scheme, NPARC, can predict the right trends in the dominant flow features and enable the designers to obtain a reasonable estimate of the inlet performance as a function of various design parameters. The present study provides a strong support that the NPARC code can be used for the design parametric studies of inlet configurations similar to that presented here and can potentially result in a significant cost as well as time reduction in the development of new engines.

ISABE-2001-1139

MULTIPLE OPERATING POINT ANALYSIS USING GENETIC ALGORITHM OPTIMISATION FOR GAS TURBINE DIAGNOSTICS

A.Gulati, D.Taylor, R.Singh

Rolls-Royce Performance Engineering University Technology Centre
School of Mechanical Engineering
Cranfield University
United Kingdom

Abstract

The paper presents a diagnostic system that uses measurements from an engine to predict shifts in component performance parameters for a relatively poorly instrumented engine.

The diagnostic problem is solved by minimising an objective function using more than one operating point for analysis. A real coded, specifically tailored Genetic Algorithm has been developed to carry out optimisation by concentrating on the faulty components. It uses a fully non-linear steady state performance simulation model. The shifts in performance parameters of the various components are quantified in the presence of instrumentation noise, measurement bias and also model inaccuracies. The parameters setting the operating point of the engine are assumed to be affected by noise and biases as well. The proposed technique takes care of the non-linear nature of gas turbine operation, smearing effect and need for more measurements than the performance parameters, which are typical problems for current techniques being used in the industry.

The technique has been developed and tested for use with a three-spool low by-pass ratio turbofan engine with a realistically limited instrumentation set. The results obtained from this model quite clearly demonstrate the accuracy and success of the diagnostics tool.

ISABE-2001-1140

Measurement of Wall Temperatures in Gas Turbine Combustors Using Thermographic Phosphor Thermometry

By

J P Feist and A L Heyes
Department of Mechanical Engineering,
Imperial College of Science Technology and Medicine,
London, UK

Abstract

The trend in gas turbine development is for increasing turbine entry temperatures due to the improvement in overall efficiency that can thereby be obtained. These temperature increases must be accompanied by the development of advanced cooling strategies making efficient use of the minimum amount of air and of protective coatings made from refractory materials. Development of cooling schemes and of predictive design codes requires accurate measurement of surface temperatures under realistic conditions.

In hot regions of gas turbines the application of conventional temperature measurement techniques is problematic. Infrared pyrometry is affected by stray light (from flames), reflections, changes in the emissivity of the observed surface and also by degeneration of optical system cleanliness during operation of the engine. Thermocouples are intrusive, suffer from high installation costs where rotating components are concerned and their coverage is limited to pre-installed points that cannot be changed during operation. However, a thermometry technique, based on the characteristics of thermographic phosphors, may enable these difficulties to be overcome.

Thermographic phosphors consist of a ceramic host matrix with a lanthanide ion dopant. When illuminated with UV light they exhibit luminescence which is temperature dependent by virtue of variations in the relative intensities of distinct emission lines or in the time constant of the exponential decay which occurs once excitation has ceased. YAG:Dy, YAG:Tb and Y_2O_3:Eu are examples and have been selected by the authors as phosphors suitable for use in high temperature gas turbine applications since each has a dynamic range extending to in excess of 1000°C. To make temperature measurements a thin layer of the phosphor is applied to the surface under investigation. The surface is then illuminated with UV light, typically from a laser, and the luminescence observed. In this way temperatures can be measured anywhere in the coated region of the surface and, since emission from the phosphor occurs at discreet wavelengths associated with the various emission lines, the output can easily be distinguished from background radiation from flames for example.

The authors have calibrated the response of a range of phosphors and demonstrated that a useful dynamic response exists up to at least 1400°C. Furthermore to demonstrate the utility of the technique the surface temperature was measured over a small region within a model gas turbine combustor.

The combustor consists of a single sector of a research combustor designed by Rolls Royce which was operated, in the laboratory, under atmospheric conditions. It is a reverse flow type combustor typical of the sort used in helicopter engines and is fuelled via an airblast atomiser using liquid kerosene. For surface temperature measurements the combustor was operated at a simulated take-off condition where previous experiments have shown the average exit plane temperature to be approximately 1450°C. For the experiments a small region of the combustor wall, in line with the fuel injector and just below the primary cooling holes, was coated with a layer of Y_2O_3:Eu. To measure the temperature, an area approximately 8mm by 8mm was interrogated in a point wise manner at 0.5 mm intervals using a pulsed Nd:YAG laser and the lifetime decay response mode. The measured temperature distribution clearly shows the position of the cooling holes which are associated with regions of low temperature. However, there is no evidence of cold streaks on the surface associated with an attached flow of cooling air emanating from the cooling holes.

ISABE-2001-1142

USE OF PRESSURE SENSITIVE PAINT FOR DIAGNOSTICS IN TURBOMACHINERY FLOWS WITH SHOCKS

Jan Lepicovsky *
QSS Group, NASA Glenn Research Center
Cleveland, OH 44135
and
Timothy J. Bencic **
NASA Glenn Research Center, Cleveland, OH 44135

Abstract

The technology of pressure sensitive paint (PSP) is well established in external aerodynamics. In internal flows in narrow channels and in turbomachinery cascades, however, there are still unresolved problems. In particular, the internal flows with complex shock structures inside highly curved channels present a challenge. It is not always easy and straightforward to distinguish between true signals and 'ghost' images due to multiple internal reflections in narrow channels. To address some of the problems, investigations were first carried out in a narrow supersonic channel of Mach number 2.5. A single wedge or a combination of two wedges were used to generate a complex shock wave structure in the flow. The experience gained in a small supersonic channel was used for surface pressure measurements on the stator vane of a supersonic throughflow fan. The experimental results for several fan operating conditions are shown in a concise form, including performance map points, midspan static tap pressure distributions, and vane suction side pressure fields. Finally, the PSP technique was used in the NASA transonic flutter cascade to compliment flow visualization data and to acquire backwall pressure fields to assess the cascade flow periodicity. A summary of shortcomings of the pressure sensitive paint technology for internal flow application and lessons learned are presented in the conclusion of the paper.

ISABE-2001-1143

INFRA RED RADIATION OF CIRCULAR AND COAXIAL JETS

by

Srinath S Heragu*, K V L Rao, B N Raghunandan@**

ABSTRACT

A novel scheme has been applied for the prediction of IR signals from circular and coaxial jets. It is a comprehensive scheme that deals with the IR radiation emanating from an engine exhaust and incident on an arbitrarily located sensor. Existing codes have been modified and adapted for flow simulation inside nozzles and simulation of circular and coaxial jets. The radiosities of the discrete panels forming a diffuse and gray nozzle surface have been determined. The nozzle surface radiosities and radiation from gas inside the nozzle have been used as boundary conditions on the jet radiation. The narrow band model has been used to model the radiation from the gas inside the nozzle and the jet. Geometric modeling techniques have been developed to identify and isolate nozzle surface panels and gas columns of the nozzle and jet in the determination of radiation signals incident on the sensor. The scheme has been validated and typical results have been generated for IR radiation from circular and coaxial jets.

ISABE-2001-1144

Testing Active Control of Combustion in a Bluff-Body Burner by Means of a Numerical Experiment

by

Eugenio Giacomazzi
ENEA - C. R. Casaccia
Sec. ING-TECO, S. P. 081
Via Anguillarese, 301
00060 Santa Maria di Galeria
Rome, ITALY

and

Claudio Bruno
Department of Mechanics and Aeronautics
University of Rome "La Sapienza"
Rome, ITALY

ABSTRACT

In this paper we propose a strategy to control combustion in an axisymmetric bluff-body burner, operated at 10 atm, by means of an auxiliary pulsed air jet injected by a duct located in the bluff-body itself. We model the non-stationary flow using the complete Navier-Stokes equations solved by using our reactive LES code, in which the so called Fractal Model (FM) for treating turbulence and combustion is employed. The pulsed air jet operation changes flame features: the pulsed jet creates two stable vortices and a hot zone anchoring the flame, while two larger vortices stretch and shorten periodically the flame; the flame main pulsation frequency is 795 Hz; the flame predicted is shorter, continuous and less thick compared to the case without controlling jet; unburnt fuel is lower. We performed also thermal NO analysis, but the emission level for this burner is too low to be significant.

Key Words: combustion, control, LES, unsteady

ISABE-2001-1147

COMBUSTION INSTABILITIES IN GAS TURBINE AFTERBURNER

K.Ashirvadam, S.Kishore Kumar, A.P.Haran
Afterburner & Exhaust System
Gas Turbine Research Establishment
Bangalore, INDIA

and

P.J.Paul
Department of Aerospace Engineering
Indian Institute of Science
Bangalore, INDIA

ABSTRACT

One of the important areas which requires considerable time and attention in the development of military engines with afterburners is the combustion instability and its attenuation. Instability results from coupling between the unsteady heat release rate and the fluid perturbation of the system. Through this coupling, energy is supplied by the combustion to sustain the oscillations. For the oscillations to decay damping built into the system should be sufficiently large to dissipate the oscillatory energy more rapidly than it is supplied by combustion. Combustion instability is characterized by chamber pressure oscillations although the frequency and amplitude of these oscillations vary with the type of instability viz., low or high frequency types. As these oscillations lead to increased shell temperature and structural failure, they have to be suppressed. This is achieved by using the perforated liner located at a distance away from the rigid wall casing. This paper attempts to explain the physics of combustion instability and present the analysis of wave propagation characteristics of an expansion chamber with a stationary medium without any heat addition. This is the first step in the analysis before introducing complexities like 'mean flow in the chamber, heat addition and perforations in the chamber wall'.

Key Words: Combustion Instabilities, Afterburners

ISABE-2001-1148

SIMULATION STUDIES ON AN AERO ENGINE AFTERBURNER CONTROL SYSTEM

S.Kathiresan, B.K.Garudachar, S.R.Balakrishnan
Gas Turbine Research Establishment
Bangalore, India

ABSTRACT

A simulation study has been carried out on the afterburner control system of a gas turbine engine as a part of design and development and this paper narrates the details of the study. It includes the requirements of the system, description of the system, modeling of various elements, software developed for simulation, validation of model and results at different operating conditions. It also includes the system design modification for holding the last controlled flow value even under the controller failure which is one of the critical requirements of the engine under consideration.

ISABE-2001-1149

Effect of Atomizer Geometry on the Performance of Simplex Fuel Atomizer

J. Xue, M. A. Jog, and S. M. Jeng
University of Cincinnati, Cincinnati, OH 45221-0072
and M. A. Benjamin[d]
Parker Hannifin Corporation, Mentor, OH 44060

Abstract

In this paper, the effect of atomizer geometry on the flow in simplex atomizers and atomizer performance is numerically investigated. A computational model based on the Arbitrary-Lagrangian-Eulerian method has been used. The effect of changes in four non-dimensional geometric parameters on the atomizer performance is studied. These geometric parameters are - the atomizer constant (ratio of inlet slot area to the product of swirl chamber diameter and exit orifice diameter), the length to diameter ratio of the swirl chamber and that of the orifice, and the spin chamber to orifice diameter ratio. The variation in the atomizer performance is obtained keeping the pressure drop across the atomizer constant and is presented in terms of the dimensionless liquid film thickness at exit of the orifice, spray cone half angle and discharge coefficient. Results indicate that the atomizer constant is the most dominant parameter among the four geometric parameters studied here. With the atomizer constant(K) increasing from 0.1 to 0.6, dimensionless film thickness at exit increases by about 0.15; the discharge coefficient increases by about 0.2; and the spray cone half angle decreases by about 20%. Both dimensionless film thickness at the exit and discharge coefficient increase, and spray cone half angle decreases monotonously with increase in L_s/D_s. Under the conditions considered in this study, with variation in l_o/d_o, a minimum in the dimensionless film thickness is at $l_o/d_o = 0.75$. Also, t^* decreases dramatically for small values of l_o/d_o and then increases slightly. With increasing l_o/d_o, both discharge coefficient and spray cone half angle decrease sharply at small values of l_o/d_o and then decrease gradually. With increasing D_s/d_o, the dimensionless film thickness at the exit first decreases, then increases for D_s/d_o greater than 4.5. However the dimensional film thickness at the exit decreases monotonically. C_d decreases with increasing D_s/d_o, eventually becoming almost constant at larger values of D_s/d_o. Spray cone half angle decreases with increasing D_s/d_o.

Key words: Atomization, Computational Fluid Dynamics, Fuel Injection.

ISABE-2001-1151

THE INFLUENCE OF DUCT GEOMETRY ON THE RADIAL VELOCITY COMPONENT OF A FUEL INJECTOR BOUNDARY CONDITION.

C. Batchelor-Wylam, R A Hicks, C.W. Wilson.,
Centre for Aerospace Technology, Defence Evaluation and Research Agency,
Farnborough, United Kingdom, GU14 0LS

Abstract

To optimise the design of gas turbine combustion chambers it is important to understand the aerodynamics of the combustor's primary zone. Computational fluid dynamics (CFD) can be used to gain insight into these mixing processes. Although 3-D CFD models of combustion systems can be solved, it is common to model the combustion chamber separately, with suitable boundary conditions. Recent work at DERA has developed design rules for deriving the aerodynamic boundary condition for an airblast fuel injector. These rules utilise velocity profiles defined at the exit of each individual swirler, and include a radial component. However, the influence of swirl duct geometry on the radial velocity component is not allowed for. A 2-D CFD model of a simple duct has been used to derive a relationship between duct geometry and the value of radial velocity experienced by the swirler. This relationship, when combined with the existing design rules can be used to calculate the inlet boundary conditions for a 3-D combustion chamber model.

Key words

Fuel injector, CFD, boundary conditions

Analysis of Spray Quality with Varying Geometrical Sizes in an Airblast Atomizrer.

S.N. Sridhara*
BMS Evening College of Engineering
Bull Temple Road, Bangalore-560 050, INDIA
and
B.N. Raghunandan♣
Indian Institute of Science,
Bangalore-560012
INDIA

Abstract:

The performance of a plainjet airblast atomizer is evaluated in terms of jet breakup modes and Satuer Mean Diameter (SMD) at varying sizes of air nozzle. Different photographic techniques have been adopted to study the details of modes of jet breakup at varied nozzle geometry. Malvern Master Sizer-χ, a laser light scattering instrument is used for the measurement of mean dropsize of the spray. At a given air-to-liquid mass ratio, increasing air nozzle diameter is shown to drastically alter the mode of jet breakup and the resultant spray structure. A simple correlation to predict the mean dropsize of the spray in terms of air-liquid momentum ratio irrespective of the nozzle size is proposed.

* Senior Lecturer, Department of Mechanical Engineering
♣Professor, Department of Aerospace Engineering

ISABE-2001-1153

Combustion Chamber Operating Condition Influence On Droplet Shape Evolution For Multi-Component Fuels

Ola. M.I. Rashed, Badia Hafez

Aerospace Department
Faculty of Engineering
Cairo University
Giza, EGYPT

Extended Abstract

Direct fuel injection is encountered in a wide range of applications such as diesel engines, jet engines and gas turbines. Depending on the system operating conditions the formation of droplets and their evaporation pattern dictates the overall combustor behavior. The changes in droplet shape during its lifetime can induce combustion problems. Recently it has been established that studying the vaporization process can lead to the understanding of the combustion instabilities plaguing liquid fuel combustion chambers. A parametric study of the combustion chamber operating condition effect on droplet shape evolution for multi-component fuel sprays is presented. The developed mathematical program models the exchange processes for mass, momentum and energy between the liquid and gas phase of vaporizing liquid droplets that vary in shape according to the operating conditions. This model accounts also for the variable thermo-physical properties, surface regression, transient droplet heating, slip, as well as the internal liquid circulation. The different operating condition effects on the droplet shape evolution are examined. By increasing the combustion chamber temperature for low-pressure cases the droplet shapes into a crescent until it vanishes. On the other hand at increased pressures depression is formed on both sides of the droplet until separation occurs, and the newly formed droplet evaporates further until vanishing. At higher pressures the droplet experiences its evaporation in an explosive manner, which may cause combustion instabilities. This developed program provides a tool for the evaluation of the droplet evaporation important for the modeling of direct fuel injection systems.

ISABE-2001-1154

Calculating Flow Field Characteristics of a Turbulent Round Jet Subjected to Vortex Generating Jets

Nabil H. Mostafa
Mechanical Power Department, College of Engineering
Zagazig University, Zagazig, 44519, Egypt.
E-mail: nmostafa@vt.edu

Extended Abstract

A computing of a turbulent jet subject to fluidic excitation and it's effect on the jet evolution was conducted. The jet was subject to vortex generating jets, VGJ, placed at the exit of the main jet nozzle exit to generate Swirl Jet as shown in figure 1. The main jet had a R_{en} about 17000, and the VGJ to main jet momentum ratio were 0.055, 0.095 or 0.286. The velocity ratio between the VGJs and main jet ($CR = C_{vgj}/C_{jet}$) was varied up to unity. The activation of VGJ was tangential (injection angle, α=0), (α=45 degree) or (α=90 degree).

In non-reacting Swirl Jets, the dynamics of the flow field structure were computed using a three-dimensional Navier-Stokes code (CFDRC, 2000). The governing equations are discretized on a structured grid using an upwind difference scheme. The macroscopic behaviour of the jet evolution is discussed with the turbulent pressure and velocities. The transport of jet fluid is compared with the unexcited jet.

The use of Vortex Generator Jets to enhance the mixing between a turbulent jet and the surrounding fluid is shown to offer improvement and control over the jet evolution. VGJ's enhance the jet spreading angle over unexcited jet. The Swirl Jet with injection (α=45) and the higher momentum injection gives maximum higher Turbulent Kinematics Energy than the Baseline Jet by 18.5, 29.4, 38.7 or 122.6% for mr equals 0.055, 0.078, 0.095 or 0.285 respectively as shown in figure 3. With Swirl Jet excitation the high stress region located at a cylinder layer of radius 0.4 r/D and moving horizontally in a section from 2 to 3 z/D from upstream according to momentum ration. These computational data are compared with the experimental results obtained with a four-wire hot-wire velocity probe by the author (Mostafa et al,1999 and Mostafa, 2000). The results show great agreement with the experimental results at different conditions of Swirl Jet flow as compared between figure 4 and 5.

Keywords

Swirl Jet, Vortex Generating Jets, CFD, Turbulent.

REFERENCES

CFDRC, 2000, "CFD-ACE+ Theory and Users' Manuals Ver.6.4" *CFD Research Corporation*, Huntisville, Al., USA.

Mostafa, N. H., Le-Pera, S. and Vandsburger, U., 1999, - "Evaluation of Turbulence Scales in A Jet Subjected To Vortex Generating Jets" " N. H. *Forum on High speed Jet Flows, the3rd ASME/JSME Joint Fluids Engineering Conference,* San Francisco, CA.

Mostafa, N. H., 2000, "Space/Time Variation of Energy in the Jet Subjected To Vortex Generating Jets" *Forum on High speed Jet Flows VI, ASME Fluids Engineering Conference,* Boston, Massachusetts, USA.

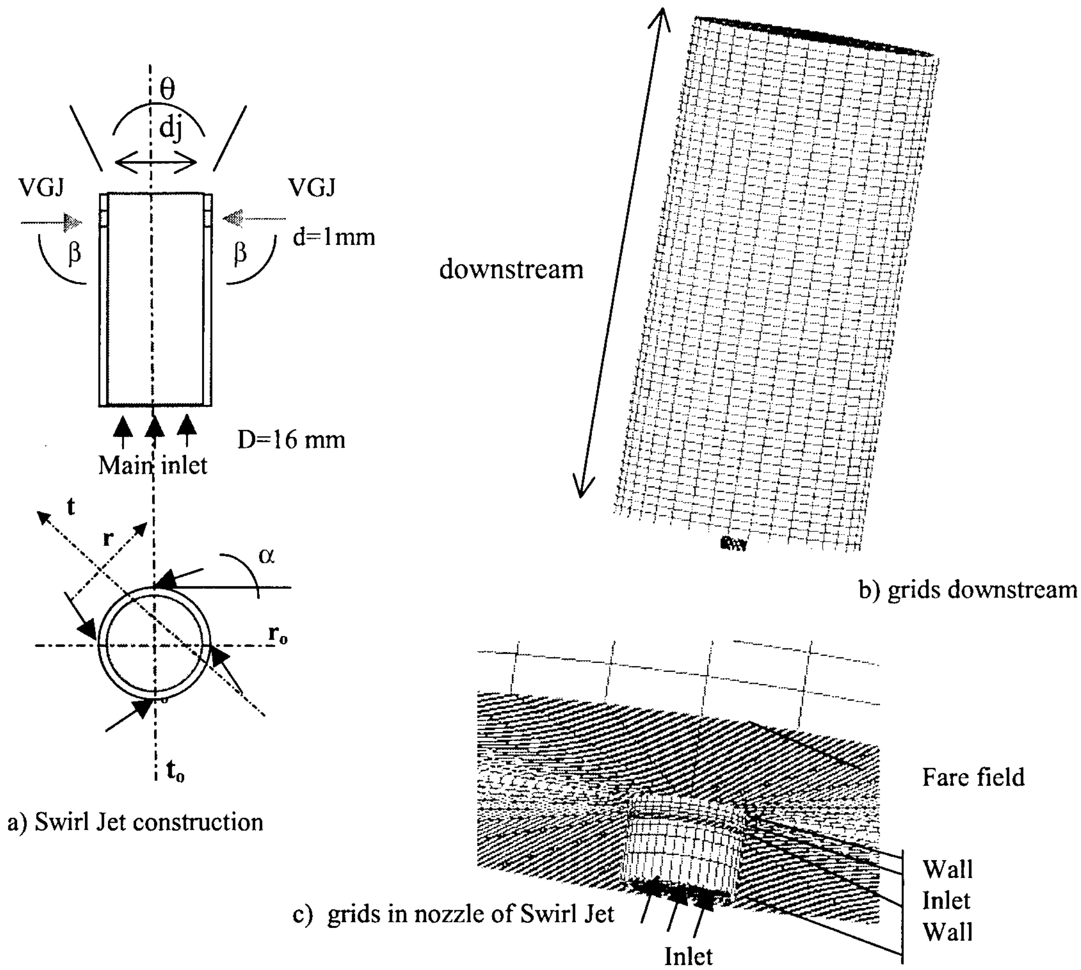


Fig. (1) grids in Swirl Jet and its downstream.

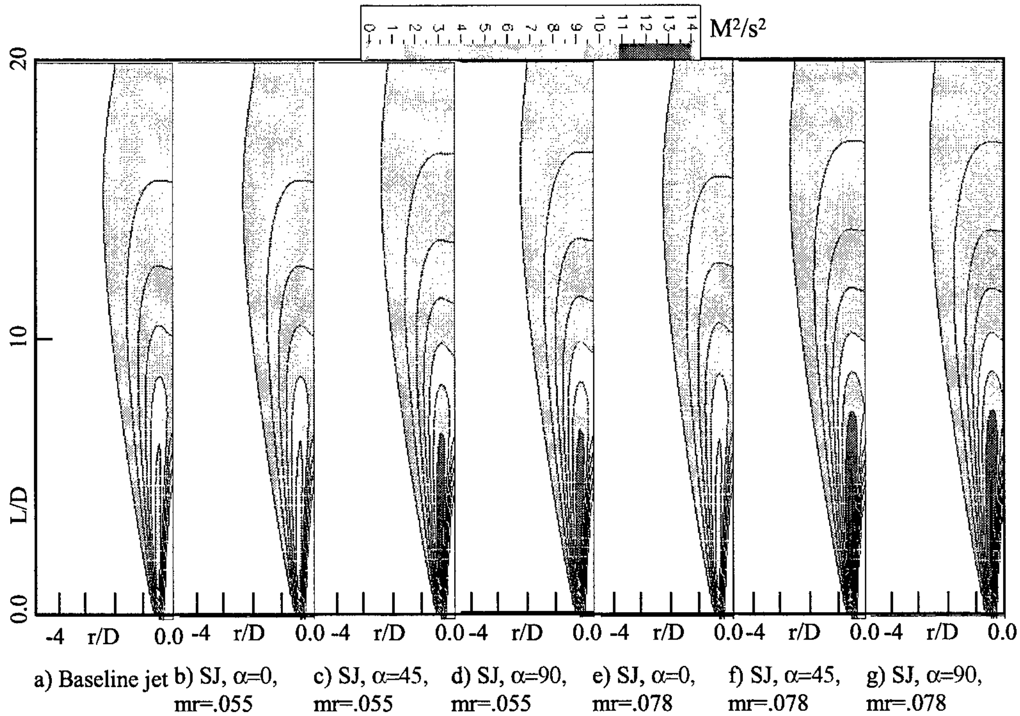


Fig. (4) Iso-contour of turbulent kinetic energy (k) for baseline jet and Swirl Jet across diametric plane.

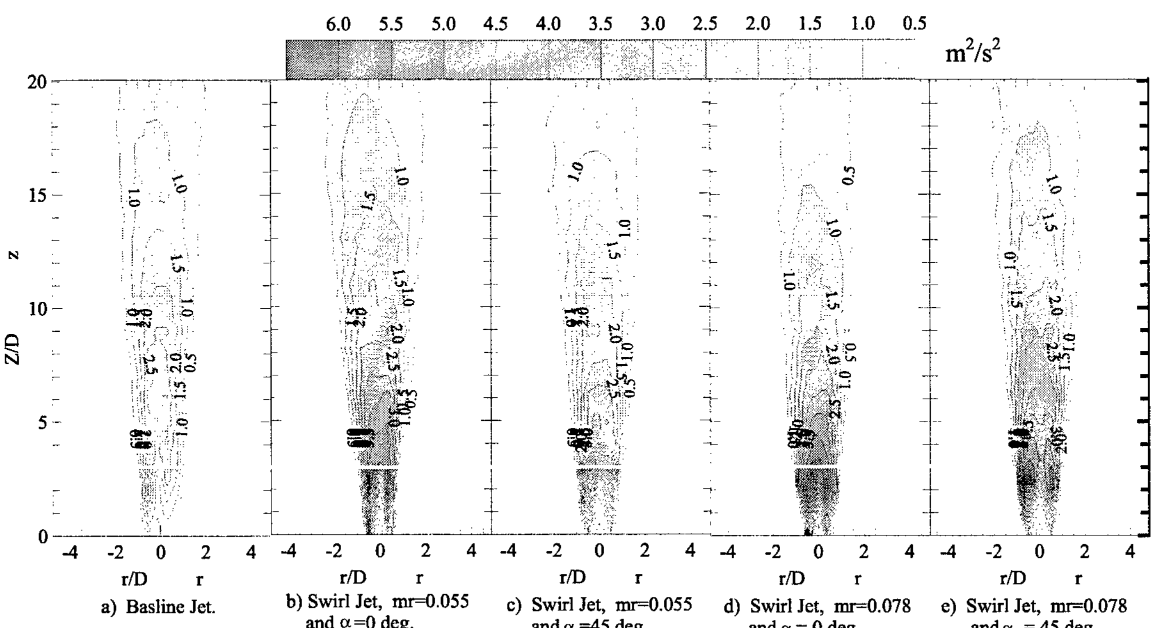


Fig. (5) Iso-contour of total turbulence energy ($U'^2+V'^2+W'^2$) for Basline Jet and Swirl Jet across diametric plane, Mostafa (2000).

A detailed experimental analysis of the flow in a highly loaded fixed compressor cascade: The ISO cascade co-operative programme on code validation.

J. Délery and G. Meauzé
ONERA - 29, avenue de la Division Leclerc - 92320 Châtillon - France

Abstract

The proposed paper presents the main experimental results obtained in the ISO Cascade tested in the ONERA S5Ch facility. This study was carried out in the frame work of a co-operative program involving Snecma, IHI and ONERA. Its aim was co constitute a specially well documented test case to validate advanced predictive methods based on the solution of the Reynolds averaged Navier-Stokes equations applied to cascade flows. The objective was establish an as complete as possible description of the flow in a fixed cascade in order to provide an experimental basis with well defined boundary conditions. The choice was done to simulate a fixed compressor cascade with only two blades operating at high subsonic Mach number in order to produce the essential physical phenomena met in a highly loaded real compressor cascade: strong compression with possible separation of the blade upper surface, important side effects, occurrence of complex vortical structures resulting from large separation, shock-wave boundary layer interactions....

The flow in the ISO cascade was thoroughly investigated for an upstream Mach number close to 0.7 with the natural turbulence level of the wind tunnel. The following measurements were executed:

- wall pressure distributions on the blades upper and lower surfaces and on the channel side walls,
- surface flow visualisation to determine the skin friction line pattern in order to precisely identify the separation lines and traces of vortical structures,
- probings by multi-hole pressure probes,
- determination of the mean velocity field and Reynolds tensor components with a three-component LDV system.

Surface flow visualisations and flow field surveys have been jointly used to establish the topology of the flow in one of the passage. A careful construction of the skin friction line pattern reveals that this patter contains several foci and separation lines indicating the existence of vortical structures which results from the separation of the flow on the blade suction surface and at the junction between the blade and the side walls.

The results obtained on this case have been used to perform a first validation operation with a code using the RANS approach along with simple turbulence models.

Key words : sccade flows, wind tunnel experiments, flow modelling, flow topology

ISABE-2001-1156

NUMERICAL SIMULATION OF 3-D-FLOW WITH RESPECT TO THE ANISOTROPIC CHARACTER OF TURBULENCE

Dieter E. Bohn* and Christian Tümmers
Institute of Steam and Gas Turbines
Aachen University of Technology
Templergraben 55
52056 Aachen, Germany

Franz Kreitmeier
ALSTOM Power
5401 Baden, Switzerland

Abstract

Numerical calculations with two different turbulence models of the 3rd stage of a four-stage turbine are presented. The calculations are done with the algebraic turbulence model of Baldwin and Lomax, and the two-equation k-ε cubic eddy viscosity model of Craft et al. The Results are compared by analyzing the secondary flow strucure in the tip clearance and cavity.

Furthermore, two different approaches for the coupling between stator and rotor are investigated: a mass averaged and the Balance-Based averaged calculation of the flow quantities in the mixing plane.

Keywords: numeric, turbulence models, Balance-Based

ISABE-2001-1157

LARGE EDDY SIMULATION OF UNSTEADY FLOWS IN TURBOMACHINERY

By

Seungbae Lee*, Akshai K. Runchal, Hooi-Joong Kim*, Jin-Wha Kim***
***INHA University**
Department of Mechanical Engineering
Inchon, KOREA 402-75120723
**** Analytic and Computational Research Inc.**
Bel Air, CA 90077
USA

Extended Abstract

Flows through a rotor of turbomachinery are highly complex, unsteady, and three-dimensional in nature. Unsteady computations are very essential and also challenging to predict performances at off-design points, e.g. losses at stall, and to provide information of source fields as an input to the FW-H equation for noise prediction. This study reports numerically simulated results by applying Large-Eddy Simulations to three major research areas of turbomachinery, which are termed as a compressor cascade, a pump inducer, an axial fan. An efficient fluid dynamics model for time-dependent, two- and three-dimensional representations of turbulent flows in turbomachineries and far-field radiation from it has been developed, incorporating large-eddy simulation and Lighthill's acoustic analogy. The subgrid-scale model employed uses a hybrid of Smagorinsky's model and mathematical deduction.

The unsteady, turbulent flows with incidence angles ranging from -408 to +208 have been computed for the compressor cascade. The oscillating separation bubbles attached to the suction surface do not modify wake flows dynamically for negative incidence angles. An incidence angle greater than 88 caused a separated vortex near the leading edge to be shed downstream and created stalling. The separated vortices in each passage do not occur simultaneously and show the typical mode of rotating instability at large incidence angles.

The onset of cavitation causes head and efficiency of the pump to be reduced significantly and generates noise. To verify and supplement the empirical formula for loading and loss coefficients at off-design points, the unsteady and complex flow-fields in inducer impellers are numerically simulated for conventional and high-swept inducers. The computed pressure distributions from single-phase/multi-phase are compared with experimental data to predict the onset of cavitation.

To study unsteady vortex structures and lift-coefficient distribution in the span-wise direction of a rotating fan blade, the flows around an axial fan of DLR with 500 rpm were also simulated and the far-field noise was exactly computed using the Ffowcs Williams and Hawkings equation with quadrupole source term. The dipole noise computed at the far-field by predicted drag and lift forces at steady state was in good agreement with experimental data and the dipole source was also found to be the major factor than other sound sources from unsteady calculation.

ISABE-2001-1159

DEVELOPMENT OF AN ADVANCED HIGH PRESSURE RATIO TRANSONIC FAN STAGE. PART-I: DESIGN AND ANALYSIS.

M.Z. Chen[+], Xu Liping[+], M.V.A.Murthy[++], M.Jayaraman[++], B.R.Pai[++], R.Prathapanayaka [++]

+ Beijing University of Aeronautics and Astronautics(BUAA), Beijing, P.R. China
++ Propulsion Division, National Aerospace Laboratories(NAL), Bangalore, India

Abstract

A high performance fan stage of pressure ratio 2.0 is being designed and developed under a joint programme between the Chinese Aeronautical Establishment (CAE) China and the National Aerospace Laboratories (NAL), Bangalore, India.. Special features of the aerodynamic design are i)forward blade sweep and lean to increase the ability to bear intake distortion ii) reverse camber fan tip to reduce losses via pre compression iii) low aspect ratio of the blades to maximize stall margin. The blade will be fabricated using laminates of Carbon/Epoxy composites with tip shroud so as to limit the blade stress and deformation. Stress analysis was carried out using MSC/NASTRAN Finite Element Package. The fan stage has undergone a series of design improvements. Comparison of typical results obtained at NAL and BUAA are shown for the final version of the fan stage TTT98-29.

ISABE-2001-1160

FAN AND PROPELLER DESIGN VIA INVERSE PROBLEM ADJOINT EQUATIONS

By

Michele Ferlauto, Angelo Iollo and Luca Zannetti
Dipartimento di Ingegneria Aeronautica e Spaziale
Politecnico di Torino
10129 Torino,
ITALY

Abstract

Efficient propulsion requires high mass flow rates with moderate velocity increments. In this respect propfans seem an efficient alternative to jet propulsion but commercial applications are rare, because of unsatisfactory aeroelastic blades behavior as well as relevant noise generation. As a preliminary effort, we consider the optimization of propfan blades for thrust, in the wake of recent papers we dedicated to turbofan design [1]. This is a first step in the direction of tuning a method for noise reduction by blades shape optimization.

While usual optimization methods [2] iterate on the geometry of a certain flow problem to determine a configuration maximizing a given objective function, we propose to iterate on inverse problems [3][4], motivated by the fact that in such formulation fluid-dynamic constraints on the solution are very easily imposed.
This is possible because the controls of the optimization process are in our method flow variables, for example pressure. Hence, the flow constraints can be imposed directly in the parameterization of the controls, with no computational expense. In the design of innovative propfans, indeed, stringent flow constraints must be imposed.
The blades of the propfan are modeled as flow surfaces of zero thickness which exert forces on the fluid flow. This approximation introduces volume forces in the compressible Euler equations, which is the model adopted for the flow. In our approach, instead of modifying the shape of the flow surfaces modeling the blades, we give the force which the blades exert on the flow, and let the geometry accommodate such distribution of forces. Then the volume force distribution itself varies based on the functional gradient, so that, thrust is maximized.

As typical applications, the thrust optimization of a ducted and an unducted configuration are presented. The preliminary applications shown are a first step towards optimization of the blade planform and load for noise reduction.

ISABE-2001-1161

Response of a Turboprop Compression System to Distorted Inlet Conditions

By

A. M. Abdel-Fattah*

Defence Science and Technology Organisation
Aeronautical and Maritime Research Laboratory
Melbourne - Australia

ABSTRACT

The results are presented of an analytical study to evaluate the response of the T56 single spool turboprop compression system to disturbed inlet flow conditions. These cover both spatial and temporal types of distortion, with emphasis on hot-gas ingestion and the rapid temperature transients found during weapons firings from military aircraft. An adapted generic code DYNTECC, which was developed by Sverdrup - AEDC, USA, was used in the current study. Both the detailed compressor geometry and individual stage characteristics form a major part of the code input. These were derived in a reverse engineering procedure using the NASA code, STGSTK. This program, which uses the stage stacking technique at mid radius, was modified at AMRL to include real flow effects that were in turn derived using yet another adapted code CASCAD.

ISABE-2001-1162

Observer Based Active Stall Avoidance For an Axial Compressor Stage with Inlet Distortion

Stefan Wagner

D. K. Hennecke

Gas Turbines and Flight Propulsion

Darmstadt University of Technology

Petersenstr. 30, D-64287 Darmstadt, Germany

email: wagner@gfa.tu-darmstadt.de

Abstract

The effect of steady and transient inlet distortions generated by deltawings of different size has been investigated experimentally. The influence on local flow properties and on the global stability behaviour of an single-stage axial compressor has been extensively examined by utilising inexpensive and robust dynamic pressure probes mounted at compressor shroud in front of the rotor, hot wire anemometers and traversable dynamic pressure probe measurements.

The experiments clearly reveal that distorted inlet flow leads to a shift of the stability line to higher mass flows. At a certain angle of attack a remarkable shift of the stability line occurs. This signifcant increase in stall inception mass flow is believed to be triggered by the periodic vortices due to boundary separation on the deltawing rather than locally high flow angles.

An observer based active stall avoidance control system using robust and inexpensive instrumentation has been designed, simulated and successfully tested. A simulation model based on experimental results on the 1-d fluid dynamics and on the 2-d stall inception dynamics is used to simulate and evaluate the controller performance. Finally the systems capabiliity of stabilising the compressor is demonstrated by several experiments and the systems robustness for steady and transient inlet distortions is assessed.

Key words: Stall Avoidance, Inlet Distortions, Axial Compressor, Active Control, Stall Modelling

ISABE-2001-1163

EFFECT OF INLET DISTORTIONS, CO-SWIRL AND COUNTER-SWIRL ON SINGLE AXIAL FAN AND CONTRA-ROTATING AXIAL FANS

Bhaskar Roy, Sq. Ldr. Hari Kumar, Amit Batra
Aerospace Engineering Department, I.I.T., Bombay

Inlet distortions are flow non-uniformities that generally result in decrease in performance and equally importantly in lowering of the operating flow range of any axial fan or compressor. This paper presents inlet flow distortion study on a single axial fan and on a set of contra-rotating twin rotor axial fans. The experiments were carried out for the following inlet flow conditions: a) clean inlet condition; b) 90° circumferential extent steady total pressure distortion; c) A combination of 90° total pressure distortion and localized 90° swirl (co-swirl and counter-swirl). The total pressure measurements were done at the inlet and exit of the axial fan/contra-stage both in the distorted and in the clean regions. At each of these axial locations measurements were taken at five radial locations along the blade span. The experimental results were analyzed to obtain distortion parameters like Distortion Index, and axial/contra-fan performance parameters like the Pressure Rise Coefficient, Flow Coefficient etc. It is observed from the experimental results that the Distortion Index is highest for the case of combination of 90° Total Pressure Distortion and 90° Localized Counter-Swirl. The combined inlet distortion is much more for the contra-rotating fan unit as compared to the single axial fan, operating at same rotational speeds. A significant outcome of the study is the fact that in spite of a higher distortion, counter-swirl produces a lower degradation in performance compared to the co-swirl, which has the lower distortion intensity. Also, the performance penalty in percentage loss of pressure rise coefficient, in the case of the contra-rotating unit is less compared to the single axial fan, in spite of the former showing higher inlet distortion index.

Keywords : INLET DISTORTIONS, DISTORTION INDEX, CO-SWIRL, COUNTER-SWIRL, CONTRA-ROTATING AXIAL FANS, PRESSURE RISE COEFFICIENT, FLOW COEFFICIENT

ISABE-2001-1165

PERFORMANCE OF MODERN STOVL FIGHTER POWERPLANTS

Roy A. Cookson, Jose Luis Herrero, Dimitris Myzithras and Pericles Pilidis

School of Engineering, Cranfield University
Cranfield, Bedford, U.K

ABSTRACT,

The aim of this paper is the description of the performance characteristics of powerplants intended for modern STOVL fighters. The intended application for the propulsion system investigated here is a combat ASTOVL aircraft. Two powerplants are examined, as they appear to be the most viable solutions for a combination of high performance, increased efficiency and STOVL capability. These powerplants are the lift-nozzle and the low pressure shaft driven lift fan. Both powerplants involve the integration of a similar core, with different approaches for providing vertical thrust. They appear to be viable solutions for a combination of high performance, good efficiency and STOVL capability.

One, the engine with lift nozzles, was composed of a gas generator with an overall pressure ratio of 28, BPR 0.6, TET 2050K at cruise conditions of 1.2 Mach number and 30,000ft. The other, the shaft driven lift fan, was a dual cycle powerplant. It was composed of a conventional cruise engine (OPR 28, TET 2050K and BPR 0.5). The core was optimised for forward cruise flight at the same conditions as the previous one. For ASTOVL, a shaft driven remote lift fan, is used to generate vertical thrust.

The net thrust needed to sustain supersonic cruise at the design point was 65kN. The higher bypass core engine of the vectored thrust engine gives it a small SFC advantage, although this advantage appears to be smaller at high flight speeds. Also, the net thrust produced by the core with the lift nozzles is slightly higher in all the off-design situations, showing 130kN against those 128kN of the lift fan configuration at SLS conditions, or the 192kN against 185kN, relatively, for augmentation at design point conditions.

Once the cruise engine performance prediction was made, the STOVL simulation was attempted. The shaft driven lift fan overall powerplant's cycle was changed, as the lift generation system acts when power is extracted from the cruise engine, to offer a total vertical thrust of 164kN, against 130kN from the vectoring lift thrust engine. This shows a clear advantage for STOVL operations, backed by the better tolerance of that powerplant to the inlet temperature increase, which simulates the hot gas ingestion. However this engine is expected to be somewhat heavier.

The performance simulation described here highlights many salient features of the engine. The scope of this study was to compare the basic performance of these novel powerplants. From these further assessments can be carried out in the future related to a more detailed understanding of these advanced powerplants.

ISABE-2001-1166

A CHALLENGE IN EVOLVING CERTIFICATION REQUIREMENTS FOR AN EXPENDABLE SMALL GAS TURBINE ENGINE FOR UNMANNED AERIAL APPLICATION - A CASE STUDY

BY

S. NARAYANASWAMY

SCIENTIST

REGIONAL CENTRE FOR MILITARY AIRWORTHINESS (ENGINES)

CEMILAC, DRDO, BANGALORE – 93

ABSTRACT

One of the keys to successful Engine Certification is to establish when the propulsion system will be in trouble. Specification codes for validation and Certification procedures, test results acceptance criteria etc. for gas turbine engines meant for all types of piloted aircraft applications are available in the open literatures. Unfortunately no such codes or even the nature of test requirements are made known to public for any type of aero - engines which are meant for unmanned aerial vehicle application, probably due to its discreet application like target drones, missiles, reconnaissance etc. The concerned Certification Agency is therefore required to evolve and stipulate a suitable validation requirements / procedures. Pragmatic stipulation of certification requirements primarily stems from the intended duty cycle, environment of operation, engine expendability, recovery techniques and also its envisaged life. The fitness for according Type Certification mainly invigorates from the out come of structural and functional abilities test results.

A deliberate effort is made through this paper to illustrate design verification significance of a small, expendable gas turbine engine meant for unmanned aerial application through a case study. The engine under discussion is a straight flow jet engine, develops 370 kgf thrust at ISA/SLS condition. The highlights are not only on the need to adopt an objective oriented structural integrity tests to screen out in service failures beside the advantage of adopting mission mix oriented test bed endurance cycle to establish the achievable level of performance and also its durability. The author also emphasis the need for making available the specification / validation procedures / techniques in the open literatures for all those expendable, short life, small power gas turbine engines also irrespective of its applications. This would not only greatly benefit all the propulsion community and more so in the case of the Certification Engineers in sharing technological knowledge enabling to configure appropriate test techniques.

BIO-DATA OF THE AUTHOR

NAME : S. NARAYANA SWAMY

DESIGNATION: SCIENTIST

ORGANISATION : REGIONAL CENTRE FOR MILITARY AIRWORTHINESS (ENGINES)
CEMILAC, DRDO, MINISTRY OF DEFENCE
BANGALORE – 560 093 , INDIA

EDUCATIONAL QUALIFICATION : B. E (MECH), M.TECH(TURBO-MACHINERY)

MEMBERSHIP MIE, M.Ae.SI

Experience : More than 20 years of experience in the field of airworthiness assurance of aero engines and its systems

Fax No. 080- 5240075

Tel No. 080- 5241428

ISABE-2001-1168

A PERFORMANCE OPTIMISATION OF ADVANCED JET ENGINES

Nanahisa Sugiyama
National Aerospace Laboratory,
Chofu, Tokyo 182-8522, Japan

Pericles Pilidis
School of Mechanical Engineering, Cranfield University,
Bedfordshire MK43 0AL, UK

ABSTRACT

The performance requirements of the propulsion systems for military aircraft and prospective supersonic transports are very challenging. These requirements demand a flexible and efficient jet engine. Modern engines incorporate advanced handling features to ensure smooth and efficient operation throughout the flight envelope. These features include variable compressor stators, variable nozzle areas and, in some cases, variable area mixers.

An advanced jet engine hereby dealt with is a two-spool, low bypass, mixed flow, supersonic turbofan engine with five control variables. An investigation was carried out to explore the sensitivity of engine performance to changes in the control parameters. The main output parameters are estimated, such as SFC and thrust and a component analysis is also carried out to show the off-design paths on the compressors.

The information shown here can then be used for optimising the performance of the jet engine. Showing clearly the effect of each variable and their interactions will enable a more straightforward optimisation exercise.

ISABE-2001-1169

Commercial Engine Architecture Selection in the Presence of Uncertainty and Evolving Requirements

Dr. Bryce Roth, Mr. Brian German, Prof. Dimitri Mavris
Aerospace Systems Design Laboratory, School of Aerospace Engineering
Georgia Institute of Technology, Atlanta, GA 30332-0150

Abstract

The objective of this paper is to discuss a few challenges foreseeable for future aircraft engine designs and briefly survey ongoing research that addresses these challenges. Emphasis is placed on methods for selecting commercial engine architectures. Four fundamental needs are identified and discussed at length: uncertainty in the design process, strategic business decisions in the context of engine design, complexity of future propulsion systems, and integration of new technologies into next-generation products. Probabilistic techniques are suggested as an analytical means to quantify the impact of uncertainty and to allow for uncertainty-mitigating decisions in the design process. Advanced engineering models in conjunction with ideas from complexity theory and game theory are a possible means of addressing the larger strategic business decisions as they pertain to architecture selection. Thermodynamic work potential methods are proposed as a basis for dealing with increased complexity. Finally, the role of technology identification, evaluation, and selection methods in engine technology studies is discussed.

Key Words
Probabilistic design, engine technologies, design uncertainty, work potential, technology evaluation

ISABE-2001-1170

Practical Pulse Detonation Engines—How Far Are They?

By

Gabriel D. Roy
Office of Naval Research
Arlington, Virginia, 22217-5660, USA

ABSTRACT

The global Research and Development community has been paying particular attention to detonation-driven engines as a candidate for air-breathing propulsion in the next generation. The simplicity with fewer moving parts, lesser fuel consumption, and ease of scalability that are associated with this engine make it highly attractive. However, there are several technical and scientific issues yet to be solved before transitioning the concept of repeated detonation into a practical pulse detonation engine. Among these, direct detonation of liquid fuels, performance, fluidic thrust vectoring, heat transfer, noise, etc are of significance. This paper addresses these issues, their implications and a few of the ways in which the Office of Naval Research propulsion program is addressing them. History has shown that there is always a time for a new technology to dawn, mature and find its place in a practical application. A carefully executed research program, properly integrated with technology development can make this maturity time shorter. The accomplishments of the various researchers involved in this program constitute the essence of this paper.

ISABE-2001-1171

FUNDAMENTAL MULTI-CYCLE STUDIES OF THE PERFORMANCE OF PULSE DETONATION ENGINES

R. J. Santoro, C. Conrad, S.-Y. Lee and S. Pal
The Pennsylvania State University
Department of Mechanical and Nuclear Engineering
University Park, PA 16802
USA
Other Contributing Authors: J. Watts

Abstract

The results from a series of detonation experiments conducted to characterize the deflagration to detonation transition (DDT) process for ethylene-air mixtures in a 45 mm square, 1.65 m long tube are described. Experiments were conducted for both single shot detonations involving quiescent mixtures as well as multi-cycle detonations involving dynamic fill. For the experiments, pressure and flame emission measurements were made to obtain the compression wave and flame speeds. In addition, OH PLIF and Schlieren imaging were applied to investigate the combustion phenomena during both deflagration and detonation. For ethylene-air propellants, strategically placed obstacles were necessary to achieve DDT. The effect of the presence of obstacles on flame acceleration was systematically investigated by changing the obstacle configuration. The parametric study of obstacle blockage ratio, spacing between obstacles, and length of the obstacle section indicated that for successful DDT, the obstacle needs to accelerate the flame to a minimum flame speed of roughly half the Chapman-Jouguet (C-J) detonation velocity. Differences in the flame and compression wave velocities demonstrated the development of a coupling mechanism as the wave propagated along the tube. A series of simultaneous Schlieren and OH PLIF images showed that the obstacle plays a major role in generating large-scale turbulence to enhance flame acceleration. Localized explosions of pockets of unburned mixture further enhanced the shock wave strength to continuously increase the flame speed. The results of this experimental study support the importance of obstacles as a means to enhance DDT and provide a potential solution for practical pulse detonation engine applications.

Key Words: deflagration, detonation, engine, transition

ISABE-2001-1172

MULTILEVEL COMPUTATIONAL STUDIES OF PULSE DETONATION ENGINES

K. Kailasanath, C. Li and G. Patnaik
Laboratory for Computational Physics and Fluid Dynamics
Naval Research Laboratory
Washington, DC 20375 USA

Abstract

Various levels of computational studies and their impact on the research and development of pulsed detonation engines are discussed in this paper. The levels range from studies using one-dimensional numerical models with comprehensive chemical kinetics to those based on multidimensional models with phenomenological submodels. The advantages and limitations of the various levels of models are illustrated using examples. The results of detailed, one-dimensional numerical simulations are generalized to identify three key factors that control performance. The inherent uncertainty in the boundary conditions used in one-dimensional simulations is invoked to explain the wide variation in propulsive performance reported in the literature. Multidimensional detonation structure studies are used to explain the observed difficulty in the initiation and transmission of detonations in ethylene-air mixtures.

Key Words: PDE, Computational Studies, Detonation Engine

ISABE-2001-1173

EFFECTS OF FUEL DISTRIBUTION ON PULSE DETONATION ENGINE **OPERATION AND PERFORMANCE**

By

Christopher M. Brophy, Jose Sinibaldi, and David W. Netzer
Naval Postgraduate School
Department of Aeronautics and Astronautics
Monterey, California 93943
USA

Abstract

The temporal and spatial distribution of fuel/oxidizer mixtures in pulse detonation engines(PDE) has been shown to have a substantial effect on the performance and stability of these systems. It has been observed that by controlling the amount of fuel/oxidizer loaded into the combustor of a pulse detonation engine, the thrust and specific impulse of the system can be substantially affected while maintaining a fixed operating frequency. Improvements in specific impulse of up to 300% have been observed with fuel/air systems operating at reduced thrust levels. The homogeneity of the fuel injection has also been shown to be extremely important in the operation of these systems. Localized fuel deficits at critical areas of the combustor have been observed to result in the failure of the detonation wave into a combustion driven shock wave situation. The fuel/oxidizer homogeneity appears to be much more critical in a PDE system than a conventional subsonic combustion systems.

ISABE-2001-1174

STUDIES OF FUEL DISTRIBUTION AND DETONATION CHEMISTRY FOR PULSE DETONATION ENGINES

J. Lasheras, B. Varatharajan, C. Varga and F. Williams
Department of Mechanical and Aerospace Engineering
University of California, San Diego
La Jolla, CA 92093-0411
USA

Abstract

Pulse detonation engines (PDE's) are promising power plants for a number of different applications. The work reported here concerns airbreathing PDE's employing liquid hydrocarbon fuels, thereby addressing a most practical application. Conditions are investigated that would be relevant, in particular, to propulsion of vehicles employing JP-10 in air, at flight Mach numbers up to Mach 5 and ambient pressures down to 0.03 bar. The problems of fuel injection, atomization, fuel distribution, detonation initiation and augmentation of detonability are addressed. Results of laboratory measurements of timed-injection spray properties in cold flow and of theoretical calculations of JP-10 and augmentor shock-tube ignition times are presented. These results provide fundamental information that can be helpful in PDE design.

Atomization experiments were carried out to demonstrate the feasibility of unsteady coaxial injection for PDE use. Droplet diameter and velocity measurements were taken at locations along the jet centerline, and averages were calculated conditioned on the phase of the injection pulse. Comparisons were made to the steady-flow case for averages over many pulses, with results indicating that the number-average diameter and the Sauter mean diameter are relatively constant throughout the unsteady behavior. Initial results suggest that this injection scheme can indeed provide the desired small droplet sizes and spatial fuel-air mixture distributions critical to detonation of JP-10 liquid fuel in PDE's.

This work shows how new concepts of atomization, fuel distribution and detonation initiation can benefit future developments of pulse-detonation engines. The aerodynamic-shear breakup was found to be ideally suited to this application and to produce the JP-10 drop-size distributions needed. Swirl was shown to be a useful means for tailoring radial and axial fuel distributions. A two-step chemical-kinetic description for initiation and propagation of JP-10 detonations in air was developed. The results can be used in future investigations of designs of pulse-detonation engines.

ISABE-2001-1176

GAS TURBINE REQUIREMENTS FOR POWER GENERATION CYCLES HAVING CO_2 SEQUESTRATION

Mr. A. J. B. Jackson, Visiting Fellow,
Prof. R. Singh, Deputy Head
(both) School of Mechanical Engineering
Cranfield University
Cranfield
Bedfordshire, UK. MK43 0AL
United Kingdom

H. Audus
IEA Greenhouse Gas R&D Programme
CRE, Stoke Orchard
Cheltenham, GL 52 4RZ
United Kingdom

ABSTRACT

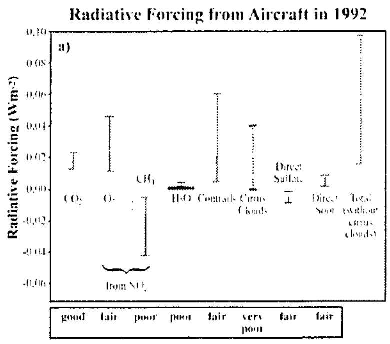
The need to reduce global warming caused by atmospheric carbon dioxide (CO_2) resulting from the combustion of hydrocarbon fuels is summarised. Five possible Options for eliminating CO_2 emissions from ground based plant powered by gas turbines are considered. The engine hardware and performance changes relative to current gas turbines burning natural gas are shown. Two of the Options burn a mixture of hydrogen and nitrogen and can be open loop. The other three are partially or wholly closed loop to concentrate the CO_2; these have CO_2 as the main working fluid, and burn natural gas, with exhaust sequestration of CO_2.

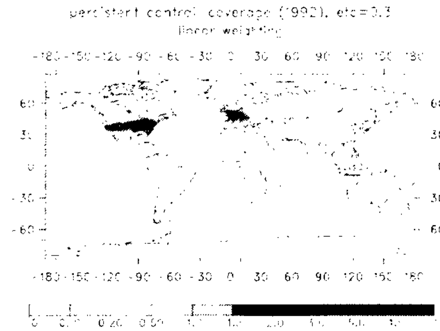
Key Words: Gas Turbine, Carbon Dioxide, Emissions, Performance.

Aircraft Particulate Emissions

Chowen Chou Wey
Army Research Laboratory/NASA Glenn Research Center at Lewis Field
21000 Brookpark Road, Cleveland, Ohio 44135, USA

Abstract





Aircraft emissions are deposited throughout the atmosphere, and at the lower stratosphere and upper troposphere they have greater potential to change ozone abundances and affect climate. There are significant uncertainties arising from the incomplete knowledge of the composition and evolution of the exhaust emissions, particularly regarding reactive trace species, particles, and their gaseous precursors.

**Figures are from the IPCC 1999 Special Report: Aviation and the Global Atmosphere*

Engine emissions measurements of primary species are available from ground tests and in-flight measurements campaigns. However, similar data for size distribution and number densities of particulate emissions and concentrations of their gaseous precursors are mostly unavailable and sophisticated measurement techniques are required for those. Furthermore, post-combustor evolution of those emissions affected by engine operating parameters has yet to be determined, and is currently not considered in engine design.

Similar situations exist for the aircraft emissions inventories and scenarios. Current inventories provide primary emissions on 1 degree latitude x 1 degree longitude x 1 km grids that can be interpolated onto the grids of chemical transport or other atmospheric models for analysis. Three-dimensional inventories and scenarios of particulate emissions have not been established since neither the data nor the methodology exists at this time to perform the task.

NASA Glenn Research Center at Lewis Field (GRC) has considered its role in answering these challenges and has been committed to strengthen its aerosol/particulate research capabilities with initial emphasis on establishing advanced measurement systems and a particulate database. Activities currently supported by the NASA Ultra Efficient Engine Technology (UEET) Program and accomplishment up-to-date will be described.

Keywords: particulate matter, aerosol

ISABE-2001-1179

Overview of " Research and Development of Environmentally Compatible Propulsion System for Next-Generation Supersonic Transport (ESPR project)

By

Yuji TOKUMASU
Director for Industrial Science and Technologies, Medical and Welfare Equipment, Industrial Science and Technology Policy and Environmental Bureau, METI

Kazuo MIYAZAWA
Director General, Applied Technology Development Department, New Energy and Industrial Technology Development Organization (NEDO)

Yoshiyuki FUJITSUNA
Managing Director, Engineering Research Association for Supersonic Transport Propulsion

Abstract

ESPR program was initiated in 1999 as the successor of HYPR program by NEDO under METI budget support in order to develop necessary technologies for the next-generation SST engine. As for project formation, 3 Japanese aircraft engine companies (IHI, KHI, MHI), 4 foreign engine companies (UTC, GE, RR, Snecma) and Japanese National Laboratories (NAL, AIST) join and work together as well as HYPR program. In ESPR program, CO2 reduction technologies, NOx reduction technologies and noise reduction technologies are especially focused as environmentally compatible technologies, which are critical to realize next-generation SST. Each target of the above 3 subjects and validation methodologies are overviewed together with some topics of interim results during initial 2 years of 5 years program.

Key Words: ESPR, next-generation SST, Noise reduction, NOx reduction, CO2 reduction

ISABE-2001-1180

The Development of LPP Low NOx Emissions Combustor Under the ESPR Programme in Japan

Hiroyuki Ninomiya, Masayoshi Kobayashi, Yasuhiro Kinoshita, Hideo Kimura
Kawasaki Heavy Industries Ltd.
Akashi Hyogo Japan

Abstract

An LPP low NOx emissions combustor aiming at 5EI.NOx has been being developed by KHI under the ESPR programme in Japan. An axially staged combustor with the LPP combustion system and a CMC liner wall is used to realize stable combustion and low NOx emissions. Fuel injector tests at the utmost 1.5MPa was conducted to evaluate fundamental characteristics such as stable range, auto-ignition , flash back, etc. Then based on the results, a multi-sector test was performed to investigate combustion aerothermal performance over the engine operation range. The conceptual design of the combustor, and fundamental test results are described here.

Key word
"LPP fuel injector", "Low NOx emissions", "Staged Combustor",

ISABE-2001-1181

Conceptual Design of ESPR CO_2 Low Emission Control

By

M. Maruyama, K. Takahashi, K. Ebina, Y. Soga
Ishikawajima-Harima Heavy Industries Co.,Ltd.
Tokyo, Japan

Abstract

This paper presents the conceptual design and preliminary analysis of CO_2 low emission control system. The control system includes stall margin control, turbine tip clearance control, secondary cooling airflow control and model-based optimum control. The simulation results show that there are mutual effects among these controls. Therefore it is necessary to optimize control parameters by means of model-based optimum control so that it can attain the best condition under various flight conditions.

Key words: ESPR, CO2 emission, model-based optimum control

ISABE-2001-1183

EULER AND NAVIER-STOKES SOLUTIONS FOR TWO-DIMENSIONAL AND AXISYMMETRIC NOZZLE FLOWS

Antonio Davis*, Hemant Kamath, Raghunandan, B.N** and Sundararajan, V.***

*** Gas Turbine Research Establishment, Bangalore 560 093, India**

**** Indian Institute of Science, Bangalore 560 012, India**

Abstract
An implicit flow solver is developed abinitio to investigate flowfields in two-dimensional and axisymmetric convergent-divergent nozzles which essentially solves Reynolds-averaged-Navier-Stokes equations in two-dimensions. Focus of the present paper is on the development and validation of this flow solver for various flow situations in two-dimensional and axisymmetric nozzle configurations and thereby attempt is made to bring out the versatility of the code developed. The flow solver is first used to predict aerodynamic performance of convergent-divergent nozzles that are typical of present day gas turbine engine exhaust nozzles that have sharp cornered throats. Certain differences between Euler and Navier-Stokes computations observed during the validation exercise are also highlighted. Prediction of shock-induced flow separation in an over-expanded two-dimensional nozzle is attempted next. Computations are also carried out to predict convective heat-transfer coefficient in an axisymmetric nozzle with cooled walls

ISABE-2001-1184

An Experimental Study on the Variable Throat Plug Nozzle for the High Speed Air-Breathing Engines

By

Hisao FUTAMURA, Hideyuki TAGUCHI, Shizuo SEKINE, Ryoji YANAGI
National Aerospace Laboratory
Aircraft Propulsion Research Center
Choufu, Tokyo 182-8522
JAPAN

Abstract

For high-speed cruising or space launching the exhaust nozzle is the final part and decisive role in generating the attainable thrust force. The demand of stable and efficient air-breathing engine operation imposes the nozzle with throat area control and expansion ratio control at the same time. Authors designed the translating flap type plug nozzle aiming wide range of throat area control and avoidance of flow separation in over expansion by simple mechanisms. The experimental results prove the good function and show agreement with CFD estimation. Additional CFD analysis cases make fine pictures at flight conditions.

Key Words

Exhaust Nozzle, Variable Geometry, Schlieren Images, CFD analysis

ISABE-2001-1185

COMPUTATION OF PLUG NOZZLE TURBULENT FLOWFIELDS

By

M. Ferlauto and R. Marsilio
Dipartimento di Ingegneria Aeronutica e Spaziale
Politecnico di Torino
Corso Duca degli Abruzzi,24
10129 TORINO - ITALY

Abstract

Experimental, analytical and numerical research on plug nozzle has been performed since the 50's in several countries. In contrast to the classical nozzle concepts plug nozzles provide, at least theoretically, a continuous altitude adaptation up to their geometrical area ratio. Two different design approaches for circular plug nozzles are possible: plug nozzle with a toroidal chamber and throat (with or without truncation), and plug nozzles with a cluster of circular bell nozzle modules or with a cluster of rectangular nozzle modu-les. The latter approach seems to be of advantage, since further losses induced by the gaps between individual modules, and the flowfield interactions downstream of the module exists can be minimized.

A Further plug nozzle configuration is the linear plug nozzle, which is foreseen for the propulsion system of the RLV X-33 concept.
Performance behaviors and flowfield development of linear plug nozzles, as function of ambient pressure is in principle similar to circular plug nozzle. However, special attention must be addressed to the influence of both end sides from where the ambient disturbs the expanding flowfield resulting in an expansion of the flow normal to the main flow direction and therefore in an effective performance loss. Especially for truncated plug nozzles, the change of wake flow behavior may strongly be influenced by the penetration of ambient pressure through both end sides. End plates, as foreseen for the linear plug nozzle of the X-33 demonstra-tor vehicle, could be used to avoid this ambient pressure penetration.

In this paper we are interested to develop a numerical procedure to solve the external nozzle flowfields in an axisymmetric plug nozzle device. The numerical approach here presented is based on a time-dependent integration of the Reynolds-averaged Navier Stokes equations together with the transport one-equation model to compute the eddy viscosity term [4]. The numerical results carried out will be validate against experimental data given in [7].

ISABE-2001-1187

STUDY OF THRUST VECTORING ON A DUCTED FAN POWERED MODEL AIRPLANE

S.P. Govinda Raju[1], V. Surendra Nath[2], Nitin Khamesra[3]

Department of Aerospace Engineering
Indian Institute of Science, Bangalore – 560012.

Abstract :

A tiny airplane named the RA with a take off weight of up to 7 kg has been developed for research in various aspects of flight mechanics of airplanes. Three prototypes – two flight worthy and one wind tunnel model - have been constructed. The RA is propelled by a piston engine powered ducted fan engine and includes the possibility of thrust vectoring in two planes. The RA is fully instrumented using an onboard flight data recorder and a number of sensors. The RA is flown by a remote pilot using radio control, but can make predetermined maneuvers using a signal injection system in any of the radio control channels for the airplane. Full wind tunnel tests of the RA have been conducted. Based on these a flight mechanics model of the RA including the effect of thrust vectoring has been constructed. Benefits of thrust vectoring have been studied using this model. Un-instrumented flights of the RA have revealed good performance and maneuverability. Instrumented flights of the RA are in progress.

ISABE-2001-1188

FUEL MIXING ENHANCEMENT BY PRE-COMBUSTION SHOCK WAVE

By

Shuji Ogawa, Byongil Choi, Kenichi Takita, Goro Masuya
Tohoku University
Department of Aeronautics and Space Engineering
Sendai, Miyagi 980-8579
Japan

Abstract

Mixing of fuel, injected into a Mach 2.3 air stream with a pseudo shock wave, was experimentally studied to investigate the enhancing effect of a pre-combustion shock wave (PSW) in dual-mode ramjet engines. The PSW was generated by back pressure control. A mixing enhancement was increased as its first shock wave went upstream relative to the injection port. Secondary flow and increase of the penetration of the injectant seem to be main mechanisms. In addition to these, the baroclinic torque was another possible force for the mixing enhancement.

Keywords: Pre-combustion Shock Wave, Fuel Mixing, Dual-Mode Ramjet

ISABE-2001-1189

COMBUSTION INSTABILITIES IN GAS TURBINE AFTERBURNER

K.Ashirvadam, S.Kishore Kumar, A.P.Haran
Afterburner & Exhaust System
Gas Turbine Research Establishment
Bangalore, INDIA

and

P.J.Paul
Department of Aerospace Engineering
Indian Institute of Science
Bangalore, INDIA

ABSTRACT

One of the important areas which requires considerable time and attention in the development of military engines with afterburners is the combustion instability and its attenuation. Instability results from coupling between the unsteady heat release rate and the fluid perturbation of the system. Through this coupling, energy is supplied by the combustion to sustain the oscillations. For the oscillations to decay damping built into the system should be sufficiently large to dissipate the oscillatory energy more rapidly than it is supplied by combustion. Combustion instability is characterized by chamber pressure oscillations although the frequency and amplitude of these oscillations vary with the type of instability viz., low or high frequency types. As these oscillations lead to increased shell temperature and structural failure, they have to be suppressed. This is achieved by using the perforated liner located at a distance away from the rigid wall casing. This paper attempts to explain the physics of combustion instability and present the analysis of wave propagation characteristics of an expansion chamber with a stationary medium without any heat addition. This is the first step in the analysis before introducing complexities like 'mean flow in the chamber, heat addition and perforations in the chamber wall'.

Key Words: Combustion Instabilities, Afterburners

ISABE-2001-1189(a)

FUEL-AIR MIXING BY CAVITY-INDUCED ACOUSTIC OSCILLATIONS IN CONFINED SUPERSONIC FLOW

B. Umesh Chandra, and S. R. Chakarvarthy
Department of Aerospace Engineering
Indian Institute of Technology – Madras, Chennai – 600 036

Abstract

An experimental investigation on mixing of a transverse jet injected into an air stream at Mach 1.5 assisted by cavity-induced acoustic oscillations has been conducted for various cavity dimensions. Acoustic pressure measurements were performed along the wall of the test-section opposite to the cavity, without and with injection of transverse jet. The results indicate the effect of injection on acoustic oscillations is minimal for deep cavities when compared with shallow cavities. Stagnation pressure measurements at the test section exit indicate marginal increase in the stagnation pressure loss due to injection of the transverse jet. Mie scattering technique was adopted to study the mixing behavior of the transversely injected flow in the main flow. Cavity dimensions for these tests were selected based on the acoustic amplitudes registered by flow past the cavities. The results, however, reveal only a slight improvement in mixing of the jet in presence of the cavity than without any cavity, for the dimensions selected for testing in this study.

ISABE-2001-1190

MIXING AND COMBUSTION ENHANCEMENT WITH LOBED MIXER

By

Yuzo INOKUCHI, Nobuhiko YAMASAKI,
Kyushu University
Department of Aeronautics and Astronautics
6-10-1 Hakozaki, Higashi-ku, Fukuoka 812-8581 JAPAN

Hiroshi KUMA,
Mitsubishi Heavy Industries, Ltd.
Nagasaki R & D Center
5-717-1 Fukahori, Nagasaki 851-0392, JAPAN

Masanobu NAMBA
Sojo University
Department of Aerospace Systems Engineering
4-22-1 Ikeda, Kumamoto 860-0082, JAPAN

Abstract

The mixing field and the combustion field downstream of the sine-shaped lobed mixer plate, which has the spanwise wavelength λ and the trailing edge height h, is numerically investigated, with parametrically changing the plate geometries and the inflow conditions. The effects of mixer plate geometries on the subsonic mixing of 30 m/sec are investigated for five cases having the different mixer geometries. Among them, the narrow pitch mixer (i.e., λ/h is small) enhances the mixing locally in the near fields due to the large contact surface area between the upper and lower flows of the mixer plate. However, the global mixing due to the large-scale streamwise vortices is suppressed in the narrow pitch mixer case and as a result, the mixing in the far fields is suppressed. The effects of velocity difference between the upper and lower flows of the mixer plate on the subsonic mixing are investigated for three cases having the different velocity ratio. The mean velocity is kept constant as 30 m/sec in these cases. The velocity differences slightly contribute to enhance the mixing in the near fields. However, the enhancement occurs only in the shear layer. On the other hand, large-scale convective mixing is suppressed and the overall mixing is suppressed in the far fields.

It is shown that for the same lobed plate mixer, the mixedness in the supersonic flow of Mach 2 is almost the same with that in the subsonic flow of 30 m/sec. That is, not the velocity but the plate geometry determines the mixedness. At last, in the reacting subsonic flow in a duct, it is shown that the flow is accelerated due to the heat release. As a result, the contribution of the streamwise vortices generated by the lobed mixer becomes weaker than the non-reacting flows.

ISABE-2001-1191

Correlation between Heat Flux Distribution and Combustion Mode in a Scramjet Combustor

Shuhei TAKAHASHI*, Shingo DEMISE*, Mitsutoshi OSHITA, and Takehisa IKEDA†**
**** Dept. Mechanical and Systems Engineering, Gifu University, JAPAN***
1-1 Yanagido, Gifu-shi, Gifu 501-1193, Japan
***** Dept. Mechanical Engineering, Nihon University, JAPAN***
1-2-1 Izumi-cho, Narashino-shi, Chiba 275-8575, Japan
† Dept. Aeronautics and Astronautics, University of Tokyo, JAPAN
7-3-1 hongo, bunkyo-ku, Tokyo 113-8656, Japan

Other Contribution Authors: Mitsuhiro TSUE, Kazunori WAKAI, Yasushige UJIIE, and Michikata KONO

Extended Abstract

A scramjet engine is expected to be operated under a wide flight Mach number range, for example, 6-10. Additionally, the scramjet engine is expected to work as a ramjet/scramjet dual-mode engine. Therefore, the entrance condition of the scramjet combustor varies in a wide range in temperature, pressure and Mach number according to the flight condition. In our previous study, it was shown that the combustion phenomena in the scramjet combustor were categorized into four modes; blow-off, weak combustion, intensive combustion, and thermal choke modes.. In the present study, heat flux distributions were measured with thermocouples experimentally in these combustion modes, respectively, and also numerical simulations were conducted to discuss the relation between the wall heat flux profiles and the combustion mode.

The experiments were carried out with Mach 2 supersonic wind tunnel, which had a vitiated heater, in the University of Tokyo. The combustor, which is rectangular and had a backward step, was connected to the wind tunnel directly. The fuel was the hydrogen gas and it was injected at the downstream of the step perpendicularly into the main flow. The static temperature of the main flow is 1145K. Wall heat flux was estimated by the temperature increasing rate of the thermocouple.

The characteristic profile of wall heat flux in each combustion mode is influenced by the flowfield in the combustor. When the shock wave system is absent (BO mode and WC mode), the heat flux profile reflects the behavior of the fuel plume. The reacting fuel plume heats the wall whereas freezing fuel plume cools the wall. When the shock wave system is present (IC mode), the heat flux profile reflects the behavior of the shock wave system. The maximum heat flux was located at the region where hot separation bubbles were generated. In the TC mode, hot separation bubbles near the step was disappeared, but high pressure level and high temperature level leads to large heat flux all over the combustor wall.

ISABE-2001-1193

AUTO-IGNITION TESTING IN A WATER-COOLED SCRAMJET COMBUSTOR

By

Takeshi Kanda, Nobuo Chinzei, Kenji Kudo, and Atsuo Murakami
National Aerospace Laboratory of Japan
Ramjet Research Center
Kakuda, Miyagi 981-1525 Japan

A water-cooled scramjet combustor was tested to make clear the effects of pressure, combustor length, fuel injection style, and wall temperature on auto-ignition performance. The auto-ignition performance degraded with increase of the airflow pressure. The ignition condition was around the second explosion limit of H-air. The presence of H_2O further retarded ignition by the large third body efficiency. It could attain in the separation between the step and the transverse fuel jet. The long combustor showed high performance, and the long separation was presumed to exist downstream of the fuel jet. Parallel fuel injection from the step base showed poor ignition ability. The combustor showed lower ignition performance than that of the no-cooled combustor. Ignition occurred near the wall, e.g., separation region, where the wall condition affect mixture significantly.

Key words : Scramjet, Auto-ignition

ISABE-2001-1194

ISOABE Conference Bangalore, India, September 2-7, 2001.

Numerical Simulation of Unsteady Rotor-Stator Interaction using Unstructured Grids

A. I. Georgiadis, D. G. Koubogiannis, K. C. Giannakoglou and K. D. Papailiou

Laboratory of Thermal Turbomachines
National Technical University of Athens
P.O.Box 64069, Athens 15710 GREECE
email : kgianna@central.ntua.gr

Abstract

A Navier-Stokes equations solver based on unstructured grids with triangular elements is used for the numerical simulation of the transonic flow through the fan stage of a counter-rotating propfan. Two rotors with 5/6 pitch ratio and a small axial gap between them are modelled at constant radius, using quasi-3D considerations. The flexibility that unstructured grids offer in discretizing complex domains is extended to multi-row computations, by generating time-varying grid zone(s) to span the inter-row gap(s). The lagged periodic boundary conditions on either blade row grids are treated through coordinate transformations, in accordance with the inclined computational plane theory. The Spalart-Allmaras one-equation turbulence model is used to effect closure. The two-rotor fan stage results are compared with measurements as well as other available numerical predictions.

ISOABE 2001-1196

ABSTRACT

Investigation of Wake-Shock Interactions in a Transonic Compressor using DPIV

S. Gogineni and J. Estevadeordal
Innovative Scientific Solutions, Inc.
Dayton, OH, USA 45440

W. Copenhaver and S. Gorrell
Air Force Research Laboratory
Wright-Patterson AFB, OH, USA 45433

Wake and shock interactions in a transonic compressor of the Compressor Aerodynamics Research Laboratory (CARL) of WPAFB was investigated using Digital Particle Image Velocimetry (DPIV). The wake generators are designed to simulate wakes of highly loaded stators without producing flow swirl. Flow visualization and instantaneous velocity distributions are presented for two spanwise locations and for closed spacing, near-stall condition. DPIV system captured the critical flow features such as wake structure, sharp change in velocity and flow direction at the bow-shock location, and the passage of the shock. The influence of the shock on the boundary layer and the wake structure as the blade passes in front of the wake generator (WG) is also demonstrated.

Key words: Turbomachinery, PIV, wake-shock interactions

ISABE-2001-1197

UNSTEADY THREE-DIMENSIONAL NAVIER-STOKES SIMULATIONS OF FAN-OGV-STRUT-PYLON INTERACTION

By

M. Unno and H. Kodama
Ishikajima-Harima Heavy Industries Co.,Ltd
229 Tonogaya, Mizuho-machi, Nishitama-Gun, Tokyo, 190-1297 Japan

O. Nozaki and T. Nishizawa
Natinal Aerospace Laboratory
7-44-1 Jindaiji-Higashi-Machi, Chofu, Tokyo, 182-8522 Japan

Abstract

This paper presents a numerical investigation of fan-OGV-strut-pylon interaction in a high bypass ratio turbofan engine. The primary objective of this study is to assess the effect of potential pressure disturbance induced by struts and pylons on the aerodynamic performance of fan rotor and OGV. Unsteady three-dimensional Navier-Stokes analyses are performed to investigate the complex flow fields due to the fan-OGV-strut-pylon interaction. Comparison between the measured and the calculated results shows that the analysis captures important features of the flow. It is found that the performance loss of fan rotor can be almost prevented when the potential pressure disturbance induced by struts and pylons is appropriately reduced.

Key words: "UNSTEADY FLOW" "THREE DIMENSIONAL FLOW" "TURBOFAN" "ROTOR-STATOR INTERACTION"

ISABE-2001-1199

NUMERICAL AND EXPERIMENTAL INVESTIGATION OF UNSTEADY FLOW INTERACTION IN A LOW PRESSURE MULTISTAGE TURBINE

Wolfgang Höhn[1]
Phone: +49 89 1489 2895
E-mail:
wolfgang.hoehn@muc.mtu.de
Fax +49 89 1489 99795
MTU AeroEngines
Department of Aeroelasticity and Acoustics
Dachauer Straße . 665
D-80995 Munich, Germany

Extended Abstract

The paper presents results of unsteady viscous flow calculations and corresponding cold flow experiments of a three stage low pressure turbine. The investigation emphasizes the study of unsteady flow interaction. A time accurate Reynolds averaged Navier-Stokes solver is applied for the computations. Turbulence is modeled using the Spalart-Allmaras one equation turbulence model. The influence of modern transition models on the unsteady flow predictions is investigated. The integration of the governing equations in time is performed with a four stage Runge-Kutta scheme, which is accelerated by a two grid method in the viscous boundary layer around the blades. At the inlet and outlet non-reflecting boundary conditions are used. The quasi 3D calculations are conducted on a stream surface around midspan allowing a varying stream tube thickness. A three stage low pressure turbine rig of a modern commercial jet engine is used for a study of the unsteady flow interaction. Besides the design point, the Reynolds number, the wheel speed and the pressure ratio are varied in the tests. The numerical method is able to capture important unsteady effects found in the experiments, i.e. unsteady transition as well as the blade row interaction. In particular, the flow field with respect to time averaged and unsteady quantities such as surface pressure, vorticity and turbulence intensity is compared with the experiments conducted in the cold air flow test rig.

ISABE-2001-1200

Optimum Initial Design of Centrifugal Compressor Stage With Genetic Algorithm

Xiao Xiang
X.L.Zhao
Institute of Engineering Thermophysics,
Chinese Academy of science,
P.O.Box 2706, Beijing, 100080, P.R.China

Abstract

In the field of turbomachinery, optimization is an important problem, and it is a focus of the research. In the last several decades, a lot of methods are employed in the compressor design, and get some success. Because the optimization function is always a implicit function and the character of the function in it's own parameter space is not too clearly, if we using traditional methods in optimization, the final result will be effected seriously by some uncertain conditions which always cause some incorrect solution. In this paper, we select the design and analysis procedures developed by NREC[1], and we use Genetic Algorithm[2] in the optimization procedure. The slip factor is obtained form Stanitz[3] or Eckert[4] equation in the calculation. By using GA, we can optimize all of 6 independent parameters in the same time, get better result than ever and avoid the defects of traditional methods.

The optimization with respect to the independent variables indicated in the input proceeds by calculating the performance of the current compressor geometry for a number of values of the variables chosen between their specified limits. In order to calculate these performance, we use GA which can deal with 6 independent variables. After a number of calculations are made, the optimum values of the variables are estimated by mathematically interpolating the results to obtain the values at which the efficiency is a maximum. This procedure of approaching the optimum is repeated until convergence is obtained. At last, the results of GA is better and more reasonable. And the overall efficiency which is obtained by GA is 0.6% higher than that by traditional method(complex).

Obviously, GA can give more accurate results in optimization, and it can give the designer full-scale characteristic of the object function, so the designer can obtain more freeness in designing, he can select more reasonable solutions form the group of results which GA give him. In a word, GA can make our designing more efficient, more accurate, and it let us escape from the heavy works

ISABE-2001-1201

MULTI-ELEMENT UNSTRUCTURED METHODOLOGY FOR TURBOMACHINERY APPLICATIONS

By

A. Hosangadi, S. Arunajatesan and V. Ahuja
Combustion Research and Flow Technology, Inc. (CRAFT Tech)
174 N. Main St., P.O. Box 1150
Dublin, PA 18917 USA
hosangad@craft-tech.com

ABSTRACT

Simulations for turbomachinery applications in various flow regimes are presented using a multi-element unstructured CFD code CRUNCH. The unstructured framework permits the generation of a contiguous grid without internal boundaries between different components of a turbomachinery system, and provides good local resolution in regions where the flow physics becomes important. The increased numerical stability resulting from these factors coupled with the parallel solution framework yields an efficient solution procedure for complex turbomachinery flows. Numerical results are presented for a transonic diffuser-volute configuration, a marine pump, and cavitation over a cylindrical headform.

Keywords: Multi-element unstructured, grid adaption, turbomachines, cavitation

ISABE-2001-1202

UNSTEADY DIFFUSER FLOW OF A TRANSONIC CENTRIFUGAL COMPRESSOR

Hartmut Krain
German Aerospace Center, DLR
Linder Hoehe
51147 Koel, Germany

Abstract

Centrifugal compressors with transonic flow are more and more used. This is mainly due to the current market situation. Today there are strong requirements for compact machines with small dimensions and low weight that can be manufactured on a low cost basis. More compactness necessitates higher levels of speed and Mach number which at least results in transonic flow conditions that are known to reduce the compressor's efficiency and flow range. The unsteady, transonic flow in the vaned diffuser inlet area of high pressure ratio centrifugal compressors is believed to cause these deficiencies.

The paper describes the unsteady flow conditions found inside the vaneless and vaned diffuser area of a high pressure ratio transonic centrifugal compressor. The compressor was designed for a rotor tip speed of 586 m/s. The vaned diffuser is very narrow (9 mm) and has a rather small exit radius ratio (r_4/r_2=1.5). Laser measurements inside the diffuser system were carried out for design and off design conditions, i.e. for different mass flow rates at design speed and for an additional operating point at lower speed (N/N_0=0.7). The Laser-Two-Focus Velocimeter available at DLR [7] was applied for the optical measurements. Locally, measurements were taken for 32 rotor/stator positions

The overall flow field is composed of measurement results taken at 5 measurement planes regularly distributed from rotor exit to the vaned diffuser exit and on 5 planes equally distributed from hub to shroud. In this paper the results obtained at mid span are discussed and presented. The unsteady effects were found to be highest in the vaneless and semi vaneless space, whereas the flow has almost steady state conditions in the vaned diffuser part. Supersonic diffuser inlet flow conditions and a very low velocity area close to the diffuser presser side were found at mid span and design speed. The low velocity area decreased with increasing mass flow rate. The inlet flow was subsonic at part load (n/n_0=0,7) and the low velocity area vanished.

ISABE-2001-1203

PRESSURE FLUCTUATIONS IN CENTRIFUGAL COMPRESSOR DIFFUSERS

R.Rajendran, S.Ramamurthy and S.Sankaranarayanan
Scientists, Propulsion Division,
National Aerospace Laboratories, Bangalore-560 037, India

Abstract

The unstable flow phenomenon like rotating stall and surge in centrifugal compressors occur at very low flow rates. These would cause unduly large vibrations in the turbomachinery components and results in mechanical failure. It is important to detect this phenomenon and avoid it by means of early warning systems. An attempt has been made in this paper to study the pressure fluctuations using high frequency miniature transducers in vane and vaneless diffusers of a centrifugal compressor.

The unstable nature of the flow is characterized. The observation from the measurements indicate the pressure fluctuations in the vane diffusers are getting amplified and carried to the exit of the diffuser whereas in the case of vaneless diffuser such a phenomena is not observed. The flow near the leading edge of diffuser plays an important role in diffuser performance. The unsteady pressure fluctuations in the diffuser are characterized in terms of instability parameter at various locations in the diffuser. The location where such an instability parameter is sensitive to the flow coefficient is identified as the diffuser throat. Through such instability parameter it is possible to provide early warning just before compressor goes into stall.

ISABE-2001-1204

OFF-DESIGN PERFORMANCE INVESTIGATIONS OF A CHANNEL WEDGE DIFFUSER
IN A SMALL CENTRIFUGAL COMPRESSOR

JongSik Oh
Korea Institutte of Machinery & Materials (KIMM)
Senior Researcher, Fluid Machinery Group
P.O. Box 101, Yusong, Taejon, Republic of Korea, *ojs@kimm.re.kr*

DaeSung Lee
Korea Aerospace Research Institute (KARI)
Head of AeroPropulsion Department
P.O. Box 113, Yusong, Taejon, Republic of Korea, *dslee@kari.re.kr*

Experimental and numerical investigations of the off-design performance of a channel wedge diffuser in a small centrifugal compressor are presented. Surge and choke conditions as well as design point are considered using somewhat limited range of experimental data and also supplementary CFD results. Some critical meanline design parameters' behavior is also investigated numerically, to rnder the basis for improved modelings in the meanline performance prediction.

In the present study, as a first step toward the vaned diffuser investigation, a channel wedge diffuser of the simplest configuration is considered. Experimental and 3D numerical investigations are respectively performed in a small centrifugal compressor to get performance characteristics at off-design operating conditions, especially in the meanline designers' point of view. Three points of operation, i.e., design, choke and surge are chosen as typical points of design and off-design performance. For a small centrifugal compressor with the channel wedge diffuser, the off-design performance of the diffuser is investigated with both experimental and numerical analysis, to draw the following conclusions. The steady interaction scheme in the present numerical method fails to predict the surge flow performance in the vaned diffuser. At choke condition, a reversed pressure pattern is generated in the diffuser inlet region. At surge condition, a considerable rise of the static pressure occurs in the diffuser inlet region. At choke condition, at the diffuser exit, a separated flow region is produced near the pressure side near hub, as a result of the secondary flow. The total pressure loss coefficient and the aerodynamic blockage up to the vane throat has the similar level at both design and choke points, but relatively about twice larger level at

surge point. For the static pressure recovery up to the vane throat, when it rises toward around 0.3, surge occurs, and its design range is found between 0.1 and 0.2. The flow angle, up to the throat, is strongly influenced by the flow rate, while the flow angle at the vane exit is determined mainly by the geometry.

ISAQBE-2001-1205

Experimental investigation of secondary air injection in a swirling flow

B.H.Krautkremer, O.Haag, D.K.Hennecke
Gas Turbines and Flight Propulsion
Darmstadt University of Technology
Petersenstr. 30
D-64287 Darmstadt, Germany
e-mail:krautkremer@gfa.tu-darmstadt.de

Abstract

As no reasonable mechanism for controlling the injection of secondary air through the liner wall of a combustion chamber exists, the only way to obtain the desired mass flow contribution is a smart design of the liner orifices. Due to the common use of swirl flows for flame stabilization purposes conventional correlations for jets in a plane cross flow can not be used for the design of the orifices. Recent experiments showed that the mainstream swirl affects the mixing efficiency and produces instabilities. Therefore an experimental investigation of a non-reacting mixing process of jets in a swirling cross flow was conducted. The emphasis of these experiments was laid on the mechanisms of jet penetration in a swirling flow field, thus providing information for different flow situations. Two different engine-near swirler types were used, with a wall attached and a wall detached flow field. The experiments were carried out in five steps of growing complexity. It became obvious that the penetration depth of a jet strongly depends on the local value of momentum-flux ratio. Zones of vanishing dynamic pressure of the mainstream which always occur at the stagnation point of recirculating flows leads to a momentum-flux ratio growing to infinity. This causes a variation of the penetration depth with time and space. Only slight variations of the combustor geometry leads to totally different conditions so that parameter variations in combustor design should be made by numerical means. Therefore the second part of this paper focuses on the capability of numerical approaches in case of the discussed complex flow field.

Keywords: combustion aerodynamics
swirl-flow
jet in crossflow
secondary air injection

ISABE-2001-1206

Vortical Dynamics and Acoustic Oscillation in Gas-Turbine Swirl-Stabilized Injectors

By

Shanwu Wang, Shih-Yang Hsieh, Vigor Yang
The Pennsylvania State University
University Park, Pennsylvania 16802
USA

Abstract

The vortical flow dynamics and acoustic response of a gas-turbine swirl-stabilized injector are investigated by means of a three-dimensional large-eddy-simulation (LES) technique. The formulation includes complete mass, momentum, and energy conservation equations. Several instability modes with well-defined frequencies, such as vortex breakdown, vortex shedding, and centrifugal instability, as well as their interactions, are observed in the flowfields. The method of proper orthogonal decomposition (POD) is employed to identify the complex flow structures. The acoustic response of the injector to impressed oscillations at the inlet is also studied over a wide range of frequency. Results show that forced oscillations have minor effects on the mean flow properties. The injector dynamics is not sensitive to the externally imposed oscillations.

Keywords: swirl flows, large-eddy-simulation, vortex breakdown, acoustic response, proper orthogonal decomposition

ISABE-2001-1208

Control of Kerosene Spray Flame Characteristics in a Swirl Burner

A. K. Gupta* and B. Habibzadeh
University of Maryland
Dept. of Mechanical Engineering
College Park, MD 20742 USA
*** akgupta@eng.umd.edu**

Abstract

The control of flame structure is one of the most important parameter since this can provide means to control flame instability, flame signatures and emission of trace pollutants. The size, velocity and distribution of droplets in the spray flame significantly affect the flame structure. Smaller droplet size in spray flames is more desirable since the smaller size is instrumental for enhancing flame stability limits. Smaller size also reduces the residence time of droplets in the hot combustion zone for vaporization and reduced emission levels. Our objective has been to use a suitable combination of burner input operational parameters for reducing the size of droplets and relocating the droplets within the flame. The relocation of droplets at specific location in the flame is important from the point of combustion instability. If the local heat release distribution in a flame can be changed then it may be possible to eliminate the combustion instability at a specific operational condition without any need for external energy, such as some active control. The usual approach adapted to control the combustion instability in flames is by some active or passive control technique. Active controls can be more effective and reliable than passive. However they are more expensive, complicated and requires incorporation of some external energy to control the instability in harsh combustion environments. The energy requirements as well as the operation of the device under harsh combustion conditions can be quite challenging. Therefore, finding means to control the flame features using some passive approach is attractive, reliable, economical since it requires less components and control devices.

There are several variables that affect the spray flame characteristics. These include fuel atomization nozzle type, atomization gas property, fuel nozzle location in the combustor, and swirl and flow distribution in the burner. A double concentric swirl burner used here has a centrally located fuel nozzle surrounding which there are two annuli. The inner and outer annuli of the burner allow independent control on the variation of swirl and flow distribution through the burner. The fuel nozzle type and location in the burner can also be changed which permits variation to the premixing distance in the burner. The facility allows examination of the effect of atomization gas and radial distribution of swirl and combustion airflow in the burner on the spray flame characteristics. A swirl blade cascade of any known blade angle setting can be placed in the two annuli of the burner to provide the desired combination of co- or counter-swirl distribution in the burner. An important feature of this facility is the capability to vary radial distribution of swirl and airflow distribution in the burner and the combustor dome geometry. A stainless steel air-assist fuel nozzle, nominally rated for 0.5 GPH, is used as the fuel nozzle. In the present study kerosene has been used as the fuel.

The effect of swirl and combustion airflow distribution in the burner on spray flame characteristics has been examined using an air-assist spray nozzle. The objective here is to control the spray flame features. The goal is to determine if one can control the distribution of droplets in the sprays, that subsequently controls the flame plume configuration. This control should allow one to transport droplets from one location in the flame to some other desired location in the flame.

In the paper results will be presented on the effect of radial distribution of swirl and distribution of combustion airflow in a burner on spray flame characteristics. Direct flame photography and droplet characteristics in terms of size, velocity and number density have been obtained for different combination of swirl and combustion airflow in the double concentric swirl burner using an air-assist fuel nozzle. The burner allowed independent variation of swirl and combustion airflow in the inner and outer annulus of the burner. Maintaining either the radial distribution of swirl or airflow distribution in the burner constant and varying the other parameter have examined the flame features. Results have been obtained for a swirl combination of $50^{\circ}/30^{\circ}$ and $50^{\circ}/-30^{\circ}$ in the inner/outer annulus of the burner, respectively. The distribution of air was changed as 25/75, 50/50 and 75/25 percent in the inner/outer annulus, respectively. The results show significantly different distribution of droplet size in the resulting spray as well as the flame plume configuration. The noise levels have been found to be very different. The results elucidate the role provided by swirl and airflow distribution in a burner to control the flame structure and combustion instability.

ISABE-2001-1209

Structure of Swirler Flame in Gas Turbine Combustor

Y.Kurosawa, S.Yoshida, T.Yamamoto and K.Suzuki
National Aerospace Laboratory of Japan
Chofu, Tokyo 182
JAPAN

Extended Abstract

The combustor of gas turbine is required to have low pollutant emission, high combustion efficiency and wide stable operating range. Swirling flow is widely used in gas turbine combustor to stabilize flame. The structure of swirl flame should be analyzed in order to meet these requirements.

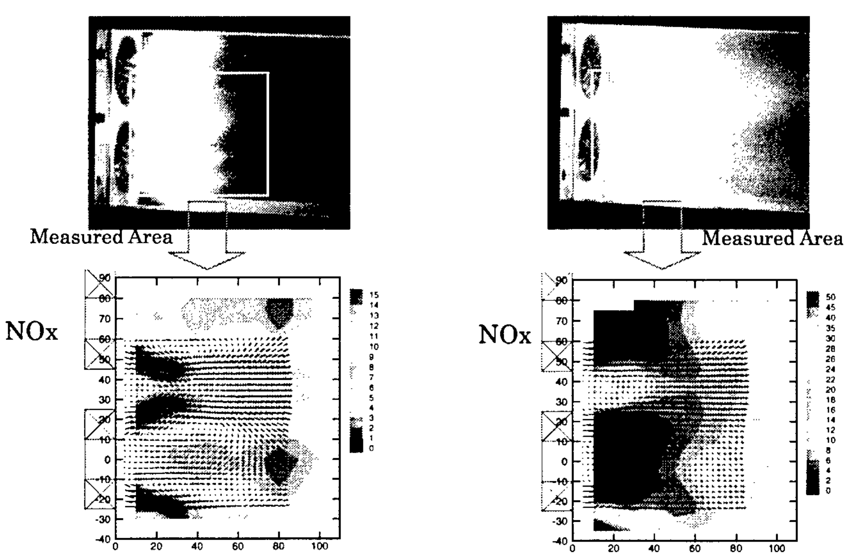
In this paper, the velocity field and the distributions of NOx, O2, CO are experimentally obtained in the combustion region to investigate the flame structures. The flow field at burning could be successfully measured by PIV. We have focused on two main factors in order to control the flame structures, how to make fuel-air mixture and how to form flow field.

Concerning fuel injection, the pre-mixed combustion is compared with the diffusion combustion. The results are presented in Fig.1 and show that the flame structures are quite different. The pre-mixed combustion is proved to be hopeful for reduction of NOx, but combustion oscillation was observed on certain condition. This problem should be solved for applying the pre-mixed combustion.

And the pre-mixed combustion with diffusion flame is also tested, because this style is often utilized in actual combustor. It should be noticed that NOx increases steeply when even small diffusion flame is added to the pre-mixed combustion.

To investigate the effect of swirling flow, the flame structures are measured for different swirlers. The reciculation zone is distorted to flat shape in strongly swirling flow. The reaction region is changed greatly by the flow field. When the distributions of chemical species are analized relating to the flow field, the combustion process can be understood more clearly.

Key Words : Swirl Flame, NOx, PIV Measurement



(a) Pre-mixed Combustion (b) Diffusion Combustion

Fig.1 Flames and NOx Distribution

ISABE-2001-1210

System Performance and Thermodynamic Cycle Analysis of Pulse Detonation Engines

By

Yuhui Wu, Fuhua Ma, and Vigor Yang
The Pennsylvania State University
University Park, Pennsylvania 16802
USA

Abstract

This work deals with the development of the system performance and thermodynamic cycle analysis of air-breathing pulse detonation engines. The model accommodates all the essential elements of an engine, including inlet, manifold/valve, injector, combustor, and nozzle. Emphasis is placed on multi-tube configurations with repetitive flow-distribution capabilities. The primary outcome is a general framework, in a form suitable for routine practical applications, for assessing the effects of all known processes on engine dynamics.

Keywords: pulse detonation engine, detonation, air-breathing propulsion

ISABE-2001-1211

MODELING OF AIR INTAKE AND ENGINE INTERACTION IN PULSE DETONATION ENGINE SYSTEMS

S. Mullagiri and C. Segal
Department of Aerospace Engineering, Mechanics and Engineering Science
University of Florida
Gainesville, Florida, U.S.A – 32611

Abstract

The effects of pulse detonation engine on supersonic air intakes have been modeled for both two-dimensional and axisymmetric configurations. The pressure perturbations at the diffuser exit have been simulated by mechanically varying the exit area resulting in a sinusoidal excitation of the backpressure, both in both in space and time. The shock oscillations arising from the fluctuating backpressure were confined within the later half of diffuser section, downstream of the throat. Experiments using Pressure Sensitive Paint (PSP) on the two-dimensional inlet yielded qualitative results on the flow field in the diffuser. The axisymmetric inlet was designed to simulate an inlet adaptable to air breathing missiles and needs further development and testing.

KEY WORDS : Supersonic intakes, Pulse Detonation Engine

ISABE-2001-1212

IMPACT OF DISSOCIATION AND SENSIBLE HEAT RELEASE ON PULSE DETONATION AND GAS TURBINE ENGINE PERFORMANCE

Louis A. Povinelli*
NASA Glenn Research Center
Cleveland, Ohio

Abstract

A thermodynamic cycle analysis of the effect of sensible heat release on the relative performance of pulse detonation and gas turbine engines is presented. Dissociation losses in the PDE are found to cause a substantial decrease in engine performance parameters.

ISABE-2001-1213

Preliminary Design of a Pulsed Detonation Based Combined Cycle Engine

Ramakanth Munipalli[*], Vijaya Shankar [†]
HyPerComp, Inc., 31255, Cedar Valley Dr., Suite #327,
Westlake Village, CA 91361, USA

Donald R. Wilson [‡], Frank K. Lu[£]
Department of Mechanical and Aerospace Engineering,
Box 19018, University of Texas at Arlington, TX 76019, USA

Abstract

A single flow path combined cycle engine using periodic detonation waves is presented here. An SSTO engine with four modes is studied:
(1) An ejector augmented pulsed detonation rocket for take off to moderate supersonic Mach numbers
(2) A normal detonation wave mode at low supersonic combustion
(3) An oblique detonation wave mode at high combustion chamber Mach numbers, and
(4) A pure Pulsed Detonation Rocket mode at high altitude.
Performance estimates based on the stream thrust approach are presented from a time averaged ideal cycle analysis with corrections from CFD results. Suggestions for performance enhancement are outlined.

Keywords: Pulsed Detonation, Combined Cycle, Multimode

---

[*] Member Technical Staff
[†] Vice-President
[‡] Professor and Chairman
[£] Professor

ISABE-2001-1214

Simulation of a Wave Rotor Pulse Detonation Engine with Integrated Ejector

M. Razi Nalim
Zuhair A. Izzy
Purdue School of Engineering and Technology
Indiana University Purdue University Indianapolis
Indianapolis, Indiana, USA.

Abstract

A new concept called wave ejector is proposed for enhanced propulsion and power applications based on pulsed detonation engine technology. The concept integrates pulsed detonation combustion process with an efficient non-steady ejector process in a wave rotor configuration. Efficient transfer of momentum and energy to bypass air attenuates the excessive velocity and temperature of detonation products. Other challenges of PDE technology can also be overcome with this approach. The technical merit of the concept is investigated, using a quasi-one-dimensional gas dynamics code to estimate augmentation of pressure gain, specific impulse, and thrust density.

ISABE-2001-1216

The Numerical Propulsion System Simulation: An Advanced Engineering Analysis Tool for Airbreathing Engines

Dr. John K. Lytle
The NASA Glenn Research Center
Cleveland, Ohio

Abstract

The advances in computing and communications technologies over the past decade have revolutionized the simulation of large-scale, complex full system simulations. This has been never more apparent than in propulsion systems. The vision for propulsion simulations at the NASA Glenn Research Center is to develop technologies that enable high fidelity, multi-disciplinary full propulsion system simulations to reduce the cost and risk associated with future aerospace vehicles. The combination of propulsion engineering and computer engineering disciplines will enable accurate, three-dimensional simulations for full aircraft engines in les than 15 hours. This capability will enable the use of these simulations in the design environment so unprecedented levels of details regarding system performance and cost will be available early in the design process before any hardware is built and tested. This vision is being implemented through the Numerical Propulsion System Simulation (NPSS). NPSS is comprised of three main elements that are required to enable large-scale complex simulations to become routine part of the design environment. Those elements are: 1. engineering applications for multi-disciplinary, multi-fidelity full system simulations, 2. a simulation environment for rapid construction of complex simulations that integrates, people, data, computing resources and analysis tools, and 3. computing platforms for low-cost, parallel, distributed processing. Multi-disciplinary analysis is necessary to capture the key physical interactions early in the design process in a more tightly coupled manner than is common practice today. Multi-fidelity analysis is necessary to minimize the size of the full system simulations while providing the designer and analyst with the detailed information required to resolve design issues. The simulation environment is one of the most critical parts of the NPSS since it directly increases the productivity of the designer and analyst by automating many of the routine tasks associated with assembling and manipulating data required to initiate complex analyses. Currently these tasks can occupy up to 50% of a designer or analyst's time. The computing environment is also a key element of increasing productivity by making available low-cost, commodity compute clusters that are geographically distributed. Analysis tools are optimized for efficient parallel execution on hundreds of processors. The NPSS capability currently consists of a US industry standard engine aerothermodynamic cycle (0-dimensional) simulation with the ability to rapidly integrate one-dimensional component analyses that can be distributed across geographically distributed Team members and computing resources. Applications can be built and executed using web-based interfaces. A major US aircraft engine manufacturer has estimated that the new object-oriented architecture in NPSS will result in a 55% reduction in the time to perform engine simulations throughout the product life cycle. The paper will describe the technologies used to enable these capabilities such as object-oriented software design, interface protocols that facilitate communication across distributed objects, personal computer cluster high speed interconnects and operating system software, and middleware that makes collection and management of distributed computing resources transparent to the user. The paper will also describe the progress toward three-dimensional, multi-disciplinary full engine simulations that will be capable of execution in less than 15 hours.

ISABE-2001-1217

COMBUSTION SUB-SYSTEM MULTIDISCIPLINARY ANALYSIS AND OPTIMIZATION USING A NETWORK-CENTRIC APPROACH 2001-1217

Charles E. Seeley, Venkat Tangirala, Ray Kolonay
General Electric Corporate Research and Development
Engineering Mechanics Lab
One Research Circle
Schenectady, NY 12301

Michael Bailey
General Electric Aircraft Engines
One Neuman Way
Evendale OH

ABSTRACT

FIPER (Federated Intelligent Product EnviRonment) is a new network-centric engineering environment which enables design optimization of complex turbomachinery systems. In this paper, the architecture of the FIPER environment is discussed. Concepts such as the Product Control Structure (PCS), Product Assembly (PA), Context Model Architecture (CMA) and Linked Model Environment (LME) are introduced. A practical application, based on a combustor subsystem, is presented. Detailed context models are developed for conjugate heat transfer, system mechanical response, detailed FEA stress analysis and system dynamics. Results are presented which demonstrate the effectiveness of combustion sub-system design and analysis using the FIPER environment.

Optimization, FIPER, distributed computing, combustion

ISABE-2001-1218

Use of CFD for the Design of Small Turbomachinery Components

G. Sieros, Research Assistant and K. D. Papailiou, Professor
Lab of Thermal Turbomachines
National Technical University of Athens

Abstract

It is nowadays well established that CFD may help considerably in supporting aerothermodynamic design of engines and apparatuses in which complex flow patterns are observed. This paper presents such a case, in which an alternative design approach is considered, in order to increase the performance of radial flow compressors.

Special emphasis is placed on the problems encountered in the design of small size components, where the introduction of advanced computational tools is relatively recent. The emphasis is on the way in which the available computational tools can be used in order to improve the performance of a centrifugal compressor. All the stages of the design procedure are described in the full paper. The design starts with a simple 1-D optimization, giving the overall dimensions of the impeller and diffuser. The exact shape of the impeller and diffuser blades are then determined through an iterative process, where it is our goal to ensure a beneficial blade loading, that will result in a separation-free flow. Typical velocity distributions on the impeller and diffuser blades are shown in figure 1 and figure 2 respectively

As a final step a 3-D Navier-Stokes calculation is performed, to check the validity of the various assumptions made in the preceding steps. The results indicate that the design that was finally chosen ensures a separartion-free flow, with a low level of losses. A test compressor was built, based on these results, and the first experimental results show very promising performance levels.

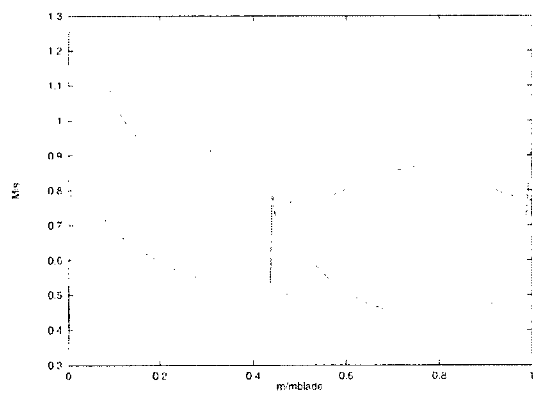


Figure 1: The velocity distribution along the blades at the tip when the calculation ended. No flow separation can be observed.

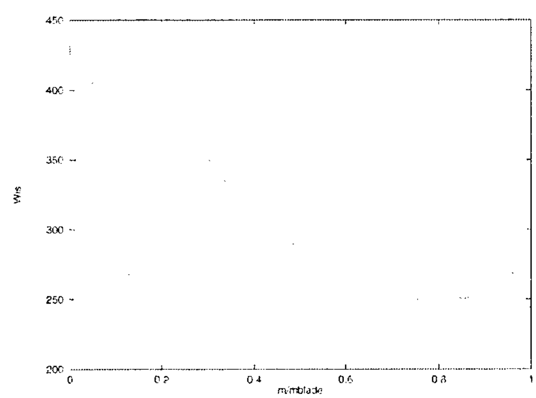


Figure 2: The isentropic velocity distribution along the diffuser blade surfaces, computed by the NTUA Navier-Stokes solver

Keywords: Compressor Design, Viscous flows, blade design

ISABE-2001-1219

NON-DETERMINISTIC MULTI-DISCIPLINARY OPTIMIZATION OF COMPOSITE ENGINE STRUCTURES

Galib H. Abumeri[1] and Surendra N. Singhal[2]
QSS Group Inc.
21000 Brookpark Road
Cleveland, OH 44135

And

Christos C. Chamis[3]
NASA Glenn Research Center
21000 Brookpark Road
Cleveland, OH 44135

Abstract

A computational simulation method is presented for Non-Deterministic Multidisciplinary Optimization of engine composite materials and structures. A hypothetical engine duct made with ceramic matrix composites (CMC) is evaluated probabilistically in the presence of combined thermo-mechanical loading. The structure is tailored by quantifying the uncertainties in all relevant design variables such as fabrication, material, and loading parameters. The probabilistic sensitivities are used to select critical design variables for optimization. In this paper, two approaches for non-deterministic optimization are presented. The non-deterministic minimization of combined failure stress criterion is carried out by: (1) performing probabilistic evaluation first and then optimization and (2) performing optimization first and then probabilistic evaluation. The first approach shows that the optimization feasible region can be bounded by a set of prescribed probability limits and that the optimization follows the cumulative distribution function between those limits. The second approach shows that the optimization feasible region is bounded by 0.50 and 0.999 probabilities.

ISABE-2001-1220

Influence of Blade Tip Gap Shapes on Aerodynamic Performance in Turbomachinery

By

Zhengming Wang Xicheng Jia Ruixian Cai
Institute of Engineering Thermophysics
Chinese Academy of Sciences
Beijing, China

The influence of different tip gap shapes on aerodynamic performance of an axial-flow compressor stator under the condition of both stationary and rotating hub was investigated by numerical method. The considered gap shapes include uniform, expanding and shrinking tip gaps. The computed results shows, on the same working conditions and within a certain range of gap scale, at least for the test cases in this paper, the blade row with smaller tip gap can attain higher efficiency, and for the same circumferential leakage area, the blade row with expanding tip gap was found to have higher efficiency.

Key words: tip gap shapes, aerodynamic performance, turbomachinery

ISABE-2001-1222

Rotor-Wall Clearance Effects upon Wave Rotor Passage Flow

K.Okamoto, T.Nagashima and K.Yamaguchi
Department of Aeronautics and Astronautics,
University of Tokyo
7-3-1, Hongo, Bunkyo-ku, Tokyo, JAPAN

Of the major fluid loss mechanisms in wave rotor operations, the effects of a clearance/gap between the rotor passage or "cell" and the end wall furnishing "ports" were analyzed, both experimentally and by numerical approach. The main attention was focused upon the initiation and propagation of a shock wave (primary shock wave) and its reflection back within a cell (secondary shock wave), in accordance with the gas/air breathing through the ports. When the interference amongst the neighboring cells was taken into account, the presence of leak flows through the clearance yielded remarkable changes to the wave/flow pattern within a cell.

Key words : wave rotor, shock wave, unsteady flow

ISABE-2001-1223

Role of Tip Clearance Flows on Flow Instability in Axial Flow Compressors

C. Hah
NASA Glenn Research Center, USA

D. C. Rabe
Wright-Patterson Air Force Research Laboratory, USA

Abstract

The role of tip clearance flows on flow instability in axial flow compressors has been examined computationally. The current study is aimed at understanding how the rotor flow structure changes when operating beyond the stall limit. Steady and unsteady three-dimensional Navier-Stokes analyses were performed for a single blade passage to investigate the flow structures. An unsteady flow analysis was conducted over the full annulus to detect possible rotating stall inception. Random oscillations of the passage shock and interaction between the passage shock and the tip clearance vortex were seen to produce variations in the shock location and the flow among blade passages during the stall process. This resulted in a rotating pattern in the flow field. Emmons' classic type of rotating stall, however, was not observed inside the blade passage.

AN APPROACH FOR LIFE EXTENSION OF AXIAL COMPRESSOR BLADES OF A HELICOPTER ENGINE

Mathews P. Samuel Sambasiva Rao. V Unnikrishnan.V.
Scientists, Regional Centre for Military Airworthiness (Engines)
CEMILAC, Bangalore, India.

Abstract

Aeroengine components are designed, analysed and test-validated to achieve structural integrity, reliability and durability during a specified life. Traditionally engines as a whole are cleared for the Time Between Overhaul (TBO) life with critical rotating parts such as discs having finite cyclic lives. Parts such as compressor blades, though not limited by finite cyclic lives are initially cleared for a non-finite life which are progressively extended during the operational usage. The methodology for such life extension, while being component or system specific, caters to a broad spectrum of requirements such as failure mode analysis and destructive/ non-destructive evaluation of the ex-service components to assess the degradation of property allowables and defect tolerance levels. This paper describes the analytical and experimental work carried out to assess the potential for life extension of an axial compressor blade of a helicopter engine. The approach included visual, dimensional, destructive, non-destructive examination and metallographic assessment of ex-service blades. Stress analysis was carried out after generating the 3D model of the blade on the eroded/ degraded profile/ geometry. Based on the FMECA carried out on the blades significant failure modes have been established. Tests and analysis were configured to screen out these failure modes. As HCF was found to be the significant failure mode, especially in view of the bifurcated inlet geometry and the need for using sand filters which could excite certain engine orders, the blade samples were subjected to bench fatigue testing using the incremental amplitude method. After ensuring satisfactory condition of the blades through blade bench fatigue testing, an engine assembled with the ex-service blades which have exhausted their presently cleared life, was subjected to an endurance testing for 48 hrs. By documenting the various engineering activities involved and the satisfactory service experience with the life extended blades, the paper suggests a logical step-by-step approach, which could be adapted for assessing the scope for life extension of any similar blades.

Key words: Life extension, Compressor blade, Failure modes, Fatigue, Endurance test

ISABE-2001-1227

LABYRINTH SEAL EFFECTS IN A TURBINE FLOW FIELD

By

Bum Ho Song
Seung Jin Song
School of Mechanical and Aerospace Engineering
Seoul National University
Seoul 151-742, Korea

Extended Abstract

Labyrinth seals are non-contact seals having two or more sharp sealing knives separated by large kinetic energy dissipating chambers. They are typically used in turbomachines to minimize leakage though the gap between rotating and stationary components.

The secondary flows associated with labyrinth seals have been studied in the past, and have recently reemerged as a focus of interest for engine manufacturers. Denton et al. (1976) developed a model to estimate the leakage mass fraction through labyrinth seals and validated it by comparing its predictions against measured data. Demargne & Longley (2000) investigated the interaction between the stator shroud leakage and main passage flows in compressor cascades. Also, Pfau et al. (2000) made flow measurements in a labyrinth seal cavity.

Recently, research has also been directed at the aerodynamic effects of labyrinth seals. Hunter & Manwaring's (2000) tests revealed that the seal leakage flow experienced only a minor change in circumferential momentum. Meanwhile, the blades turned the main passage flow to create a large mismatch in circumferential momentum at the seal cavity exit. To mitigate such effects, Wallis et al. (2000) tested turning devices built inside the labyrinth seal to extract work from and turn the leakage flow.

In line with these efforts, this paper presents an analytical model that describes the overall flow response in a single stage turbine induced by a finite sealing gap at the turbine rotor. Upon going through the stage, the radially uniform upstream flow is assumed to split into two streams - one associated with the seal and the other which has gone through the blades. The former is referred to as the leakage flow, and the latter is referred to as the passage flow. The passage flow is assumed to be inviscid and incompressible while the flow in the seal can be modeled as either inviscid or viscous.

The mass fraction of each stream is found as a function of turbine and seal parameters. The turbine parameters include the reaction, the loading coefficient, and the flow coefficient. Seal parameters include the sealing gap, the depth of seal gland, and the seal pitch. For the seal flow, Millsaps' labyrinth seal model (1994) has been adopted.

The predicted leakage mass fraction increases linearly with the sealing gap. This trend agrees with Denton's prediction (1976) which has been verified by experiment. In addition, the model predicts the axial momentum defect and the underturning of the leakage flow relative to the passage flow. These trends also agree with Pfau's experimental data (2000). Thus, the model is capable of predicting the kinematic effects of labyrinth seals on the turbine flow field.

Key words : Labyrinth Seal, Seal Model, Analytic Method.

ISABE-2001-1228

PERFORMANCE EVALUATION OF LEAKAGE FLOW THROUGH RADIAL LABYRINTH SEALS IN GAS TURBINE ENGINES

N. Nanjunda Rao,
Scientist – F,
Gas Turbine Research Establishment,
Bangalore-560 093, India.

V. Sethu Manavalan,
Scientist – D,
Gas Turbine Research Establishment,
Bangalore-560 093, India.

B. V. S. S. S. Prasad,
Professor,
I. I. T. Madras,
Chennai-600 036, India.

ABSTRACT

Experimental and computational investigations are carried out to estimate leakage flow rates in a static radial labyrinth seal for a gas turbine engine. Tests were conducted over a range of pressure ratios, varying from 1.00 to 1.20 and for three seal clearance values of 1.07, 0.84 and 0.66mm. The measured values of leakage flow parameter are corroborated with the results obtained from the simulations using FLUENT computer package. The flow details, viz., the streamline pattern, velocity vectors, static pressure and turbulent kinetic energy in the seal passages are presented.

ISABE-2001-1229

CH_4 fuel Steam Reforming for Enhanced Propulsion Performance

by

Francesco Cuoco and Claudio Bruno
Department of Mechanics and Aeronautics
University of Rome "La Sapienza"
Rome, ITALY.

ABSTRACT

The purpose of the present work is to predict the products of hydrocarbons steam reforming and cracking for application to future hypersonic propulsion. In order to assess the predictive capabilities of the code (DSMOKE), its results were compared with methane reforming experimental data, supplied by TDA Research Inc. The simulations were performed assuming a plug flow reactor, with three different thermal behaviours: adiabatic, isothermal and consisting of a linear T profile. Several conditions, associated or suggested by the experimental data, were explored: i. e., different pressures, methane-steam ratios, reactor outlet temperatures and different flow rates . The first step was to reproduce the TDA experimental data by means of a general purpose kinetics database. The results of this batch of simulations showed the leading role of cracking phenomena but also indicated disagreements with experiments likely to depend on the catalytic effects some alloying elements of the stainless steel piping might have. Following these considerations, another kinetics database model was developed, stressing those reactions which seem to have the stronger influence on the steam reforming mechanism. The results obtained are relatively acceptable, considering that catalytic effects of stainless steel cannot be simulated by the software tool. The effect of reformed gas on combustion was then investigated for a supersonic combustor, for which ignition delay time, flame speed and stability are key parameters. The preliminary results confirm reforming is an effective technique for a class hypersonic vehicles and particularly for cruisers.

Key Words: Endothermic Fuels, CH_4 Reforming, Cracking, Hypersonic Propulsion.

ISABE-2001-1230

MagnetoHydroDynamic Coupled Ramjet Propulsion System: A Perspective

CLAUDIO BRUNO *
University of Rome
Rome, Italy

PAUL A. CZYSZ †
Parks College
Saint Louis University
St. Louis, Missouri

Abstract

The goal of this paper is to summarize and put into perspective recent information and analyses of the so-called AJAX integrated propulsion system concept. The initial attempt was by Bruno, Czysz and Murthy and a Parks College Senior Undergraduate Design Team. A second paper followed in 1998. As additional analyses were done and information became available, a better understanding of the AJAX system was gained. However there remain some system elements not yet satisfactorily described.

To recap history, beginning in 1990 articles about a new long-range aircraft that cruised at hypersonic speeds, named AJAX. Its reported propulsion system employed an integrated Magneto-Plasma-Chemical Engine employing a coupled (MHD) generator/accelerator. MHD is actually not a new concept. Using available literature and discussions with Russian and Ukrainian citizens, a first principle analysis[1,2] of the system was performed to determine if the concept provided a real advantage. Working with citizens of Italy and the United States, that had visited Russia, the areas of incomplete information were sufficiently filled-in for a second report[3].

This paper focuses on the impact that new information and analyses have on our understanding of the system. The AJAX system is a total energy management system that utilizes available energy (normally not recovered as useful work) to drive an electric hydrocarbon fuel reforming process, an energy by-pass propulsion system that reduces the cycle entropy rise and a beam directed energy device of unspecified design. The control of very large energy flows bypassed from the propulsion system to the directed energy system, over short time periods remains a significant challenge. The apparent focus of this system is as a long-range hypersonic cruise vehicle not a space launcher.

The authors wish to acknowledge the assistance of Professor Mark A. Prelas, Department of Nuclear Engineering, University of Missouri – Columbia. He has toured a number of the Russian nuclear facilities and provided first hand knowledge of ionization devices. The authors also wish to acknowledge the help in calculating ionization effects by Dr. P. Battistoni and Dr. L. Petrizzi of the Italian Nuclear Agency, ENEA.

ISABE-2001-1231

A PERSPECTIVE ON A COMBINED MAGNETO-HYDRODYNAMIC-SCRAMJET ENGINE

Unmeel B. Mehta, NASA Ames Research Center
Moffett Field, CA 94035
and
David W. Bogdanoff and Chul Park, ELORET Corporation
Sunnyvale, CA 94087

Abstract

In 1957, the prospects for magneto-aerodynamics were discussed. It was concluded that the possibility of decelerating and accelerating ionized-channel flows magneto-aerodynamically is especially interesting. This idea of flow control in channels with electro-magnetic effects was again reported in 1990 as one of the elements of the AYAKS concept for energy management in a scramjet engine. A conceptual hypersonic cruiser was developed with the AYAKS concept. A perspective on the combined magneto-hydrodynamic-scramjet propulsion cycle is presented herein. Selected recent research efforts are reviewed, and technical issues are identified.

The AYAKS energy management concept uses MHD devices with a scramjet engine. A MHD generator is used along the engine inlet to extract energy from the flow entering the inlet, and a MHD accelerator is used downstream of the combustor to add most of the extracted energy back into the nozzle flow. Under certain conditions with MHD devices, this energy management scheme is expected to improve the performance of the scramjet engine.

The magnet system, the MHD accelerator, and the ionization of air are the critical technologies for realizing the potential of the combined MHD-scramjet engine. Aspects of the ionization technology are discussed herein, and results of system analyses are presented to indicate the potential of the combined MHD-scramjet engine.

When equilibrium ionization is used, strong shock-compression is needed upstream of the MHD generator channel, thus producing considerable drag. When nonequilibrium ionization is used, an ionizer, usually an external power source, is required.

In the technical literature two theoretical approaches for investigating the effect of nonequilibrium ionization have been proposed: without electron-vibration coupling and with electron-vibration coupling. These approaches are discussed herein. The viability of nonequilibrium ionization in the MHD generator channel of a combined MHD-scramjet engine for hypersonic aircraft application is not certain. Moreover, cold-airflow MHD generators with electron beams for ionization to the ionization fraction on the order of 10^{-6} are unlikely to be feasible or practical. The electron-vibration energy exchange must be accounted in the presence of nitrogen when exchanges are collision-dominated or the ionization fraction is on the order of 10^{-4} and the electron temperature is approximately 3,000 K or higher.

In spite of the drag penalty imposed by equilibrium ionization, quasi-one-dimensional system analyses, including friction losses, chemical kinetics, electron-vibration coupling (for nonequilibrium ionization), and real-gas effects, lead to the conclusion that equilibrium ionization is preferable to nonequilibrium ionization for improving the scramjet performance. A combined MHD-scramjet engine does offer an advantage over the conventional scramjet engine, that is, the non-MHD scramjet engine in narrow ranges of speeds and design parameters. The combined engine appears advantageous for hypersonic cruiser application and for the first stage of a two-stage-to-orbit application.

ISABE-2001-1232

EXPERIMENTAL DEMONSTRATION OF MAGNETO-HYDRODYNAMIC (MHD) ACCELERATION—FACILITY AND CONDUCTIVITY MEASUREMENTS

David W. Bogdanoff and Chul Park, ELORET Corporation
Sunnyvale, CA 94087
and
Unmeel B. Mehta, NASA Ames Research Center
Moffett Field, CA 94035

Abstract

It is well known that an ionized gas flow can be decelerated or accelerated, at least in principle, by the application of a magnetic field and an electric current, thereby generating electrical power or converting electrical power to the kinetic energy of the gas flow. There have been many experiments in which electrical power generation has successfully been achieved by MHD means. However, relatively few experiments have been made to date for the reverse case of achieving gas acceleration by the MHD means. MHD acceleration has several potential aerospace applications. The first is to improve the performance of hypersonic scramjet engines for space launch and cruise vehicles; the second is to improve the performance of a high enthalpy wind tunnel; and the third is to control a hypersonic vehicle.

This research report starts with a description of the NASA Ames EAST facility, which was or will be used for all studies described herein. This is followed by a discussion of the design of the MHD accelerator. The technique for generating the required magnetic field is outlined. The range of operating conditions for the MHD accelerator were selected to provide a MHD stream momentum increase of 20 – 50 percent. The diagnostic techniques, electrical and optical, are discussed. The most important aspects of the planned research program are the measurement of flow acceleration and Joule heating in the channel. Special attention will be paid to shorting in the boundary layers parallel to the current due to locally high load factors and energy loss in the perpendicular boundary layers due to cathode and anode fall heating.

We next discuss two earlier experimental studies to demonstrate techniques which will be used to measure conductivity and electron number density in the MHD channel. The first of these studies involved the measurement of the bulk conductivity of unseeded air shock heated to temperatures, pressures and electron number densities suitable for MHD accelerators. From the bulk (or pseudo-) conductivities, estimates of the electrode fall voltage drops are obtained. We show that current densities up to 300 A/cm^2 can be passed through air between two electrodes without any sign of breakdown.

The second study involved the measurement of the bulk and true conductivity and electron number density of shock heated hydrogen. While the shock conditions and the working gas for these studies are perhaps not suitable for an MHD accelerator, the techniques developed for true conductivity and electron number density measurements will be used in the MHD accelerator studies.

Valuable knowledge gained during the conductivity studies has been applied to develop a stable of diagnostic techniques to be used in the MHD accelerator studies. This includes test time determination using monochromators and total light measurements, current transformer and phase-compensated voltage divider techniques, spectroscopic analysis with the ICCD camera and the high-speed IMACON framing camera techniques.

ISABE-2001-1232

Presenting Author:
Unmeel B. Mehta

Dr. Mehta is currently a Division Scientist in the Space Technology Division at NASA Ames Research Center (ARC). He obtained his Ph.D. in 1972 from Illinois Institute of Technology, Chicago. He began working at ARC in 1973. He is an Associate Fellow of AIAA.

As a researcher, he has authored papers/reports on magnetohydrodynamics energy bypass scramjet propulsion, computational fluid dynamics (CFD), credible computations, transonic aerodynamics, dynamic stall, large-eddy simulations, and on aerodynamic computations for a high-speed magnetic flight system. As a technologist, he has contributed on strategy for developing airbreathing aerospace planes an on next generation space transportation. From June 1986 to July 1993, Dr. Mehta dealt with technology, development, and programmatic issues related to CFD for the National Aero-Space Plane (NASP) Program. He led the NASP CFD Technical Support Team (1988-1991). From March 1992 to July 1993, Mehta was detailed at the NASP National Program Office, Palmdale, California. He provided assistance to programmatic activities including program definition and advanced launch systems.

He extended contributed toward the activities of the AIAA Computational Fluid Dynamics Committee on Standards (CFD/CoS). In 1999, he formed the JANNAF Modeling and Simulation Subcommittee.

Dr. Mehta has received awards and recognition including the NASA Certificate of Appreciation for leadership and NASP Gene Zara Award. Mehta contributed to books titled *Hypersonic Airbreathing Propulsion, Handbook of Fluid Dynamics and Fluid Machinery*, and *Knowledge-Based Problem Solving*. Authored or co-authored 51 technical papers or reports. Presented numerous invited presentations at national and international conferences and institutions, including a keynote address in a plenary session at the ASME Fluids Engineering Division Summer Meeting, July 1996.

ISABE-2001-1233

COLD FLOW ANALYSIS OF AN AERO-ENGINE GAS TURBINE COMBUSTOR CONFIGURATION

H.S.Muralidhara[+], T.R.Shembharkar[+], B.R.Pai[+]
P.S.Ramanujam[*], G.N.Jayaprakash[*], P.N.Pandey[*]

+ Scientists, Propulsion Division, National Aerospace Laboratories, Bangalore.
* Scientists, Combustion Group, Gas Turbine Research Establishment, Bangalore.

Abstract

The present work is related to the computation of cold flow in a typical aero-engine gas turbine annular combustor and validation of the predictions with the experimental results. The primary motivation of the study was to simulate the complex 3-D turbulent flow in the complete combustor system, starting from the compressor exit to the turbine inlet, in a coupled manner. A general-purpose CFD code PHOENICS was employed for this purpose. The complex geometry was simulated by appropriate BFC grid. The poracity concept was used to simulate the holes smaller than the local grid size. The flow through the swirler was modelled by extra source terms in flow equations. Three-dimensional Reynolds averaged Navier-Stokes equations were solved with the standard k-ε turbulence model. Computations were carried out for three flow conditions for the combustor and the predictions were compared with experimental results like mass flow distribution and total pressure loss.

The predictions indicate that the flow is biased in all the three cases towards the outer annulus of the combustor; the outer annulus is receiving about 4% more mass flow than the inner annulus and there is no appreciable change in core mass flow. The difference between predicted and measured mass flow split varies from case to case, but on an overall basis, it may be said that around 2.5% deviation from experimental values is observed for mass flow split along outer annulus, core and inner annulus. There is very good agreement for total pressure loss. The deviation in total pressure loss is generally found to be less that 1%. The predictions indicate that the flow splits along outer annulus, core and inner annulus are relatively insensitive to change in inlet Mach number, unlike the total pressure loss which increases with increase in Mach number.

Key Words : combustor modelling, annular combustor

ISABE-2001-1234

TURBINE CASCADE OPTIMIZATION USING AN EULER COUPLED GENETIC ALGORITHM

Ozhan Oksuz and I. Sinan Akmandor
Middle East Technical University
06531 Ankara, TURKEY

Abstract

The optimal aerodynamic performance of a turbine rotor cascade is found. An Euler algorithm and a genetic algorithm are coupled within an automated optimization loop. The multi-parameter objective function is based on the tangential blade force. For given rotation speed, inlet Mach number and blade chord, the flow inlet and exit angles, the blade thickness and the blade pitch are optimized. Effects of uniform crossover, creep mutation, different random number seeds, population size and number of children per pair of parents on the performance of the genetic algorithm are studied. It is shown that the maximum tangential force is achieved for a higher flow turning, a wider pitch and a thicker cascade.

Keywords: Genetic algorithm application, cascade optimization

ISABE-2001-1235

Numerically Efficient Global Stability and Control Analysis of Flow through Axial Compressors

N. Ananthkrishnan and Vinayak Walimbe
Applied Nonlinear Dynamics and Chaos Lab,
Department of Aerospace Engineering,
Indian Institute of Technology (Bombay), Powai, Mumbai 400076, India

Abstract

The continuation and bifurcation method is presented as an efficient tool for the global stability and control analysis of the axial compressor flow dynamics. The use of the bifurcation method to both predict and control the global stability characteristics of the Moore-Greitzer axial compressor dynamic model is demonstrated. The need for care in using and interpreting results from bifurcation analyses is highlighted with examples.

ANALYSIS OF CONICAL DETONATION WAVES

By

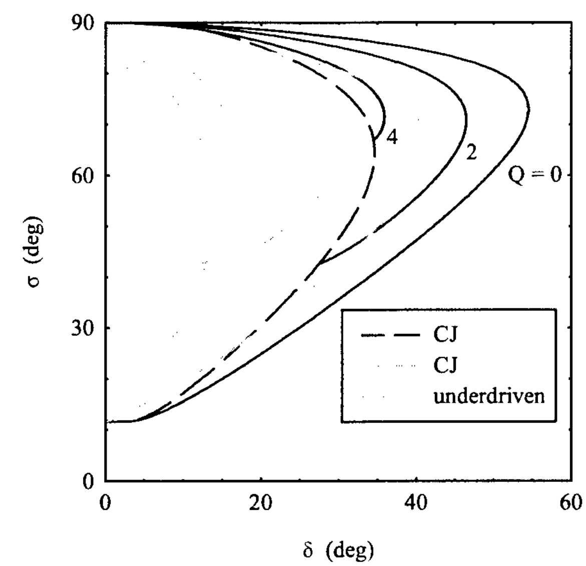
Silvano R. Saretto, Dario Pastrone and Giuseppe Bussi
Politecnico di Torino
Dipartimento di Energetica
10129 Torino, ITALY

Abstract

Since 1940's many different concepts to make use of detonation waves as applied to hypersonic airbreathing propulsion have been explored. As far as standing wave engines are concerned, oblique detonation wave engines (ODWE) have been devised as an effective mean to propel trans-atmospheric vehicles. Although several papers have been presented on these topics, the proposed analysis assumed a planar geometry, and consequently, most of the attention has been focused on planar oblique detonation waves. Moreover, although some insight into conical detonation waves has been gained through experiments and the use of CFD codes, to the best of the authors' knowledge a simple model like those used for planar oblique detonation waves has not been developed yet.

The detonation wave is treated as a discontinuity, and frictionless, non-reacting, non-adiabatic, steady flow with constant specific heat ratio is assumed. Combining the steady conservation equations of mass, momentum and energy, an equation that relates the upstream Mach number, the second Damkoehler parameter, the detonation wave angle and the flow deflection angle can be written. This equation is then used along with the governing equations for a flow past a cone to obtain the detonation wave angle for any given value of the upstream Mach number, the second Damkoehler parameter and cone semi-angle. From a detailed analysis of the properties of the differential equations that govern the behavior of a conical flow, it is found that underdriven detonation waves are not possible.

Results are presented for different values of the upstream Mach number and of the heat release parameter, and compared with those obtained in similar conditions for a planar detonation wave. Analogous to the adiabatic case, it is found that cones of larger angle than wedges can be used before the wave detaches from the cone tip. As an example in Fig. 1 the detonation wave angle σ is plotted against the wedge/cone angle δ in the planar (gray lines) and the conical (black lines) cases for different values of the second Damkoehler parameter Q and for an upstream Mach number equal to 5. The lines for $Q=0$ correspond to the planar and the conical adiabatic shock.



Keywords: DETONATIONS - CONICAL DETONATIONS - ODWE

ISABE-2001-1237

EXPERIMENTAL STUDIES ON AIR TURBO RAMJET ENGINES FOR HYPERSONIC FLIGHT VEHICLE (PART III)

Nobuo TSUJIKADO
Dept of Aeronautics and Astronautics, Faculty of Engineering, Tokai University
1117 Kitakaname, Hiratsuka, Kanagawa 259-1292 JAPAN

Masatoshi KOSHIMAE, Rikiya ISHIKAWA and Kazuki KITAHARA
Gas Turbine Div. Kawasaki Heavy Industries, Ltd.
1-1 Kawasaki-cho, Akashi, Hyogo 673-8666 JAPAN

Abstract

In order to develop air turbo ramjet engine for super- and hyper-sonic flight vehicle, Kawasaki Heavy Industries Ltd. Have been carrying out basic experimental studies on components consist of the experimentally assembled engine. The experimental engine was composed of fan driven by top arranged turbine with high temperature and high pressure combustion gas produced by liquid/solid rocket engine type gas generator, supersonic air intake module, ramjet combustion chamber including mixer/flame holder with ignition torch burner and exit thrust nozzle.

Using the engine, flight simulation tests M=3 at 12,000 altitudes were satisfactorily conducted at ramjet test facility by direct air intake connection. As result of tests, there are rooms for improvement on several important components such as: 1) Promotion of activity and configuration of decomposition catalyst of hydrogen peroxide in gas generator. 2) Refinement of combination of mixer and flame holder with high intensity ignition torch burner for ramjet combustor.

To simplify the ATR system, solid propellant rocket type gas generator were tested on the rocket test stand combined with experimental engine. Preliminary investigation on hybrid rocket type gas generator for simplifying the system also conducted

ISABE-2001-1238

EXPERIMENTAL INVESTIGATION OF A BYPASS-REGULATED SOLID FUEL RAMJET COMBUSTOR IN VARIABLE FLIGHT CONDITIONS

Deborah Pelosi-Pinhas and Alon Gany
Faculty of Aerospace Engineering
Technion – Israel Institute of Technology
Haifa, Israel

Abstract

A comprehensive experimental investigation aimed at the examination of a solid fuel ramjet (SFRJ) regulation concept using an air-division valve has been conducted. By controlling the ratio between the port and bypass flows, one can maintain a desired working state over a wide range of flight conditions. A general regulation law based on maintaining a constant fuel-to-air ratio has been tested. The experimental strategy was to isolate key parameters influencing the fuel regression rate, testing them one at a time, keeping the other parameters constant. Experiments aimed at airflow rate, total air temperature and pressure, as well as port diameter effects simulated a broad flight envelope, from Mach 1.5 to Mach 4.7 and from sea level to 13 km altitude, showing a very good agreement with the theory and demonstrating the feasibility, effectiveness and characteristics of the air-division valve regulation technique in SFRJ motors.

Key words : Solid fuel ramjet, air-division valve, bypass airflow, regulation

ISABE-2001-1239

ENERGY CHARACTERISTICS ESTIMATES FOR LAUNCH-VEHICLES WITH METHANE RAMJET ENGINES*

Valeriy I. Timoshenko, Igor. S. Belotserkovets, Valeriy P. Galinsky

The Institute of technical mechanics of National Academy of Sciences and National space agency of Ukraine
15, Leshko-Popel`St, Dniepropetrovsk 49005
UKRAINE

Abstract

The calculation model of a trajectory of injecting launch vehicles with various types of engines is described. The capability of use thermochemical fuel conversion in rocket-ramjet engines is considered. The results of calculation consumption and power characteristics of engines and parameters of the trajectory of injecting are given. The advantage of use ramjet engine at the second step of the three-step launch vehicle of an light-class started with the carrier-airplane on a S-figurative trajectory for injecting payload into low earth orbits is shown.

ISABE-2001-1240

COMPLEX NOZZLES FOR REACTIVE FLOW AND AFTEND INTEGRATION

J.D. Hoffman and S.N.B. Murthy
Purdue University, West Lafayette, IN USA
And
P.J. Ortwerth
Boeing Airplane Company, Rocketdyne Division

Abstract

In the development of hypersonic flight vehicles, the aftend configuration including thrust nozzles involves a large number of parameters changes in which affect substantially the mechanical and thermal loads, and moments. A package of tools has been developed, based on three-dimensional, rotational, chemically reactive flow method of characteristics, that is fast, reliable, accurate, and easily applied, for the design and analysis of such configurations. The methodology is also applicable to the development of fuel injection ramps and struts. Illustrative examples of application are included.

Keywords: nozzles; directional loads; three-dimensional flows; reactive flows; aftend integration.

ISABEW-2001-1241

Aeromechanical Design of Advanced Engine Compressors

R. Walther, J. Frischbier, R. Selmeier

MTU Aero Engines

D - 80976 München

Germany

Abstract:

In addition to ongoing efforts to reduce fuel consumption, chemical and noise pollution levels, future improvements for commercial engines mainly focus on economic manufacturing and maintenance of engine components. As a consequence, for future compressors, compact designs are pursued featuring low stage and blade counts. In order to meet the high aerodynamic and mechanical challenges, advanced compressor design is facing the physical limits supported by powerful and reliable analytical tools.

Integral designs such as BLISKs and BLINGs in combination with advanced fiber reinforced materials – i.e. carbon fiber reinforced plastics and metal matrix composites – offer additional potential for substantial weight reductions without compromising the component life. MTU successfully pursues these key technologies with analytical in depth investigations, material testing as well as rig and spin testing of BLISK and BLING components.

In the present paper the application of advanced aeromechanical tools used for the design of recent military and commercial compressors are described and comparisons with actual test results are presented.

Key Words: Advanced compressor technology, Blisk, Bling, Fiber reinforced materials

ISABE-2001-1243

COMPARATIVE ANALYSIS OF BLADE MODE SHAPE INFLUENCE ON FLUTTER OF TWO-DIMENSIONAL TURBINE BLADES.

Olga V. Tchernycheva, Torsten H. Fransson
Royal Institute of Technology
S -100 44 Stockholm, Sweden

Robert E. Kielb
Duke University
Durham, NC 27708-0300 USA

John Barter
GE Aircraft Engines
Cincinnati, OH 45215-1988 USA

Abstract

Recent investigations identified the mode shape as a significant design parameter for determining the aerodynamic stability of a low pressure turbine blade. The present study considers four significantly different low pressure turbine blade geometries and draws overall conclusions of blade mode shape influence on flutter.

A parametrical study summarising the effect of the reduced frequency and the mode shape is carried out with a linearized inviscid flow model and under pure subsonic flow through a cascade. Contour plots of the critical (flutter) reduced frequency as a function of torsion axis location are compared for the four turbine profiles. The results provide essential information for influencing the aerodynamic stability by controlling torsion axis location.

The agreement for the flutter reduced frequency is similar for all profiles. The main directions of the highest stability and instability regions are identified for torsion- as well as for bending-dominated mode shapes.

Key words: aeroelasticity, aerodynamic stability, mode shape, flutter.

DYNAMIC ANALYSIS OF ADVANCED HIGH SPEED GEARS WITH RESPECT TO INTERNAL EXCITATION

Zdenek. Dolezal
Aeronautical Research and Test Institute (VZLU, a.s.),
Beranovych 130, 199 05 Prague 9 – Letnany
Czech Republic

Abstract

With increased requirements for high speed, heavy loads and light weights in gear design, the prediction and control of dynamic forces and gear vibrations have become critically important. But despite many investigations on the internal excitation in the system composed of a geared pair from different object of study, many aspects of the dynamics in involute spur gear are still not satisfactorily understood. The models proposed by several investigators show considerable variations, not only in the effects included, but also in the basic assumptions and simplification conditions. The complications usually arise from the inclusion of various effect such as damping and internal excitation, generated in tooth mesh. In most of studies models include constant linear damping, fixed-periodic variable gear mesh stiffness, fixed-periodic variable transmission ratio and fix contact length in the mesh.

The initial mathematical non-linear model, proposed by the author for simulation, prediction and control of dynamic forces or gear vibration with six degrees of freedom, represents the single stage spur gear with low contact or high contact ratio. The pinion and wheel are modelled as two rigid cylinders, which are flexibly (isotropic or anisotropic and with damping) supported and predominantly connected along the contact line. The contact loss and tooth backlash are in the model included too.

The gear mesh stiffness and the transmission ratio are calculated as functions of transmitted load, gear profile errors, gear profile modification, gear tooth deflections, as well as the position of contact. An important feature of this model is that off-line-of action contact is respected too. Due to the Hertzian compression considered in the computation of tooth mesh stiffness and above mentioned functions, the tooth mesh stiffness and the transmission ratio are not independent of dynamic load, which requires en iteration cycle for the computation of gear vibration and loads. A computer program calculates the static and dynamic loads acting on the mesh or on the gear teeth, the variation in transmission ratio (as a source of global kinematics excitation), the variation in variable-variable gear mesh stiffness (as a source of parametric vibration) and the rotation and transverse vibration of the pinion and the wheel.

The solutions of the differential equations were obtained by two different numerical integration methods. In fact the same results were obtained by Newmark integration method or by Hamming modified Predictor-Corrector integration method started by Runge–Kutte integration method. While the integration methods were used to integrate the differential equations of motion, an iterative procedure was applied to solve the statically indeterminate problem of multi-tooth pair contacts, load sharing, operational contact ratio, gear mesh stiffness and transmission ratio.

ISABE-2001-1245

UNCONVENTIONAL CYCLES FOR AERO GAS TURBINE ENGINES BURNING HYDROGEN

Stefano Boggia, Anthony Jackson and Riti Singh
Cranfield University
School of Mechanical Engineering
Cranfield, MK 43OAL UK

Abstract:

One basic method of reducing CO_2 emissions is to use hydrogen as the fuel. This brings other advantages: the much higher energy content, which gives a reduction in fuel mass; the availability of a valuable heat sink, which can be used to improve cycle performance.
The present paper focuses on the use of hydrogen in aero gas turbine engines and on the exploitation of its cryogenic properties in unconventional cycles. Four novel concepts are applied to a turbofan aero engine; for each cycle the performance at take-off and at cruise is presented. An estimation of the weight and size of the engine is then made.

Key words:

Gas turbine, Engine, Hydrogen, Novel concepts

ISABE-2001-1247

DESIGN AND DEVELOPMENT OF A BIFURCATED Y-DUCT INTAKE FOR A MODERN COMBAT AIRCRAFT

R.Jolly and T.G.Pai, Aeronautical Development Agency, Bangalore
P.Jayasimha, Aircraft Research and Design Centre, HAL, Bangalore

ABSTRACT

This paper addresses design and development of a high performance wing shielded, short duct air intake for single engine fighter aircraft. A fixed geometry fuselage mounted intake with a bifurcated y-duct configuration has been selected. After initially discussing its design features like selected location, boundary layer diverters, throat sizing and duct layout, wind tunnel evaluation towards configuration freeze and establishing engine-intake compatibility are discussed. Some insight into intake flow such as "buzz" and pulse is covered. This includes correlation of rise in fuselage static pressure with buzz onset and effect of fuselage protuberances on buzz. Full scale intake pressure recovery from engine ground runs is compared with the estimates and model measurements.

ISABE-2001-1249

The problem of Vibratory Combustion in Afterburners in Turbojet Engines

Dr. A.Andreyev – Lyulka Saturn, Inc. , Moscow, Russia
Dr. V. Chepkin - Lyulka Saturn, Inc. , Moscow, Russia
V.Nesterov - Lyulka Saturn, Inc. , Moscow, Russia

Abstract

The report summarizes experience of optimizing afterburners in the AL-21F turbojets and the AL-31F turbofan in terms of processes stability relative to vibratory combustion. The main problems of combustion stability and methods of solving the task, unique methodology of defining vibratory combustion boundaries developed by Lyulka Saturn, Inc. which allow to define impact of different configuration factors on the unstable engine operation are presented in the paper. Means and devices used for afterburner optimization ensuring combustion stability and reliable engine operation as a whole are described below.

ISABE-2001-1250

EXPERIMENTAL STUDY OF FLOW IN THE CORNER REGION OF A COMPRESSOR CASCADE

LIU Huoxing CHEN Maozhang JIANG Haokang

***(Beijing University of Aeronautics & Astronautics, Beijing,*100083)**

Abstract

Turbomachinery designers are very interested in the endwall region since it is particularly dominant in modern low aspect ratio turbomachines. This region typically contributes around 1/3 of the total losses in a turbine stage. The secondary flows which are generated from the endwall boundary layers on the hub and casing of the machine significantly influence total pressure losses inside turbomachinery blade channels. The endwall region is a very complex flow which combines the effects of strong mean flow, three- dimensionality,embedded vortices, large regions of separation and turbulence, and significant pressure gradients so that the analytical techniques which can give a good prediction for the midspan are not applicable. Therefore the study of the endwall region can provide more information to alter the existing analytical methods.

The main object of present work is the corner region. The corner region is defined as the part of the flow adjacent to the endwall and the surface of blade. There are two different corner region in the endwall region,we call the region close to the suction side of the blade S corner region ,and the region close to the pressure side of the blade P corner region. The corner region of a cascade was chosen for study because it combines the interaction between the blade boundary layers and the endwall boundary layers ,the developing of the two legs of the horseshoe vortex . Based on the past numerous studies of turbine cascade passage , a detailed mean-flow structure and turbulence characteristics of flow in the corner region of a compressor cascade passage have been measured by using a three-dimensional Laser Doppler Velocimetry system. A large blade with 1000mm chord and 1000mm span was used in experiment.

The result indicated that: in the S corner region, the interaction between the blade boundary layer and endwall boundary layer thicken the blade boundary layer at downstream near the trailing edge . A strong passage vortex and a distinct horse-shoe vortex is identified on the section at 7% chord of blade(see Fig.1). With the movement to downstream, the passage vortex move to leave the suction surface of blade. The presence of the passage vortex has large effects on the behavior of the turbulence kinetic and Reynolds stress. They have maxima in the core regions of the passage vortex (Fig.2).

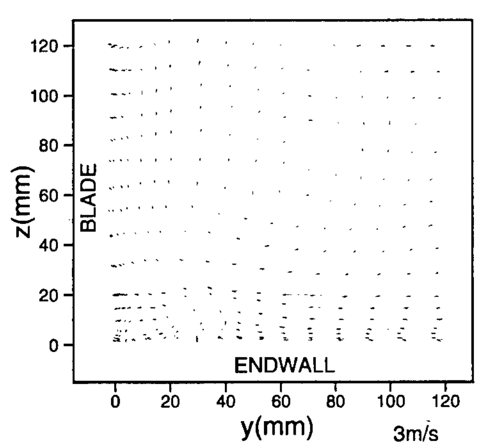


Fig.1 Secondary flow vector in suction corner

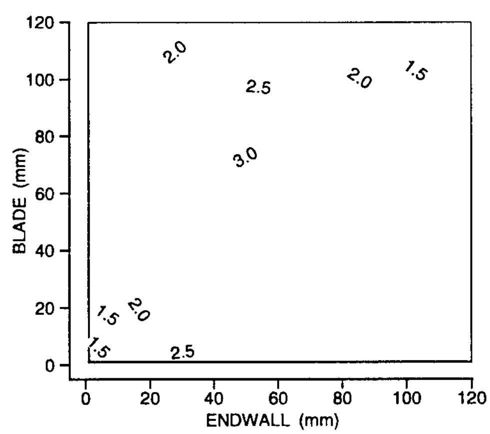


Fig.2 Turbulence kinetic in suction corner

AUTHOR INDEX

Use DTIC® Products and Services Today

http://www.dtic.mil
1-800-225-3842

Process as one —
Supplement follows
main volume.
assign same SBIN to
both

DEFENSE INFORMATION SYSTEMS AGENCY
DEFENSE TECHNICAL INFORMATION CENTER

For permission to copy or republish, contact:
American Institute of Aeronautics and Astronautics
1801 Alexander Bell Drive
Suite 500
Reston, VA 20191-4344

ISBN 1-56347-515-4

XV ISABE

XV International Symposium on Air Breathing Engines

INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

Supplementary

Book of Abstracts

Organised by the Indian National Organising Committee – XV ISABE

Bangalore, India

September 2-7 2001

XV ISABE

XV International Symposium on Air Breathing Engines

ISABE

INTERNATIONAL SOCIETY FOR AIR BREATHING ENGINES

Supplementary

Book of Abstracts

DISTRIBUTION STATEMENT A
Approved for Public Release
Distribution Unlimited

Organised by the Indian National Organising Committee – XV ISABE

Bangalore, India

September 2-7 2001

AQ F02-07-1181 b

PREFACE

The Proceedings of the ISABE, including all of the invited lectures and contributed papers, is published in every Symposium for distribution to the attendees during the Symposium. In the case of the XV ISABE, all of the manuscripts received upto the end of June, 2001, have been included in a CDROM disc attached to a Book of Abstracts.

Inspite of our best efforts, for various reasons, a number of papers arrived late. These have now been included in the Supplement to the Proceedings, consisting of a second CDROM disc and a supplementary book of abstracts.

The editorial work for the Supplement was greatly assisted by the Indian National Organizing Committee, and the material was manufactured by M/s HDDG, Bangalore, India. The ISOABE is very appreciative of this timely and valuable service.

The XV ISABE Proceedings thus consist of two discs - one main and the other marked Supplement.

Paul Waltrup
Editor

KVL Rao
Supplement Editor

August 2001

ISABE2001-1002

Indian Aeronautical Programmes and Perspectives

V K Aatre*
Defence Research and Development Organisation,
Govt. of India, New Delhi

ABSTRACT

Modern India has made great strides in various sectors of aviation technology and has remained in the forefront of all advances. Today airborne platform like combat aircrafts, helicopters, UAVs are all highly complex, integrating several sophisticated systems at considerable cost. The increased reliability and accuracy of these aerospace vehicles are mainly due to the tremendous progress in propulsion technologies, miniaturization of the electronic guidance and control systems, evolutionary improvements in engine which have resulted in greater fuel economy, and better performance characteristics. However, one of the challenging areas of today's aeronautical technology remains as air breathing engines for supersonic platforms. Air breathing hypersonic vehicles is currently under study by several countries and some astounding progress is being made globally in this area. Indian efforts in propulsion technologies are an integral part of the larger national programme in the development of aerospace systems.

The new millennium has started extremely well for Indian aerospace technology. Programmes such as Light Combat Aircraft (LCA), Pilotless Target Aircraft (LAKSHYA), UAV (NISHANT), Missiles such as PRITHVI, AGNI, AKASH, NAG and "BRAHMOS" have been successfully completed flight trials. In addition, Advanced Light Helicopter (ALH) has also been successfully designed and flight tested. In the field of Aerospace activities a range of Satellite launch vehicles including ASLV, PSLV and GSLV have been designed and deployed for several space research programme activities. In the field of air breathing engine, the KAVERI engine tailored for LCA operational requirements a small gas turbine for PTA, and a Jet Fuel Starter for LCA are in the forefront of Design and Development programme.

* Scientific Advisor to Defence Minister and Secretary, Defence R&D

ISABE 2001-1003

Indian Space Program : National and Global Contributions

K Kasturirangan*
Indian Space Research Organisation, Bangalore, India

ABSTRACT

With the dawn of independence in India in 1947 the national resources and efforts were directed to build a new social and economic order out of inherited illiteracy and poverty of a large section of the population. Since very early stage, science and technology was recognized to be a crucial apparatus for this build-up process. It is under this backdrop that the vision of Indian Space program was conceived and formulated by Dr. Vikram Sarabhai, the father of Indian Space Programme. It is hardly a surprise therefore that socio-economic concerns form the nucleus of the space endeavour.

The Indian Space Program is different. It is unique in many respects. It is very deeply routed to society and humanity. Very few space programs – if at all – can boast of this. Unlike most other space faring nations, Indian approach to space was shaped by its desire to contribute to the development of the society. Due to their enormous potential for enhancing the quality of life of common citizen, two sectors were hand picked for exploration - telecommunications and remote sensing / meteorology. These procreated two national space systems - INSAT for telecommunications and IRS for natural resources management. Once the targets were identified, two alternatives presented themselves for their realisation. The easy alternative of utilizing foreign space services either under cooperative arrangements or under commercial terms offered the charm of being quick and effective.

Indian approach to space has yielded rich dividends. India is now self reliant in space even though it does not mean producing all technological systems. Most technologies are mastered and absorbed although not all of them are put into mass production. Remarkable benefits have reached the common man in a very timely and cost-effective manner.

*Chairman – ISRO and Secretary, Department of Space.

ISABE 2001-1015

Propulsion Systems – Technology Options and Opportunities

Kota Harinarayana*
Aeronautical Development Agency, Bangalore, India

Abstract

As we look into the future, there is no substitute for fire power and mobility of aircraft and in tandem there is no substitute on the horizon for gas turbine engines as the primary propulsion system. While manned aircraft will continue to dominate the skys during the next two decades, the role of unmanned aircraft will also increase in the years to come. Every new generation fighter costs higher than the previous generation fighters, but with better performance. However, these costs are un-acceptable and unaffordable especially for developing countries. Along with avionics, propulsion system today contributes to a significant portion of this rising costs. Managerial and technological options need to be explored not only to arrest this cost price but also to reverse the trend.

* Programme Director

ISABE-2001-1028

An Assessment of Orthorhombic Aluminide Alloys for Aeroengine Applications

D. Banerjee*
Defence Metallurgical Research Laboratory
Hyderabad-500058, India

ABSTRACT

High pressure compressor stages of jet engines use nickel base alloys such as IN 718 in the last stages of the compressor operating at temperature ranges between 550C and 650C. A window of opportunity exists for significant weight savings if titanium alloys can be developed for applications in this operating temperature regime. Significant challenges in the form of burn resistance, environmental resistance and creep strength must however be overcome to exploit the specific strength of Ti alloys. While gamma aluminides show significantly better burn resistance, it is believed that the poor impact resistance of these materials can prove to be significant barrier for their use in applications where foreign object damage is likely to be a significant factor. Alternative approaches are based on the exploitation of the orthorhombic aluminide alloys. A brief history of the development of these alloys is presented, and data from recent alloy development programs is used to provide a relative assessment of these different materials.

* Director, DMRL

Air Intakes : Role, Constraints and Design

Gérard Laruelle*
EADS Launch Vehicles, Les Mureaux, France

EXTENDED ABSTRACT

The aim of this general paper is to give an overview of the problems that arise in the design of air intakes, giving many current examples from around the world.

The role of the air intake is first outlined as it concerns the engine downstream of it, bringing out the vehicle-dependent constraints for the full installation including both components. Many different flight configurations are considered: take-off, cruise, manoeuvres, landing, weather conditions, stealth, air combat. Constraints will of course also depend on the vehicle's Mach number range.

Subsonic air intakes can be divided into two categories: flush and Pitot air intakes. These are presented with their main characteristics and the design tools used for them. The design task is more complex for supersonic air intakes, and even more so for hypersonic, which is why there are two broad families of intake geometries: axisymmetrical and two-dimensional. These are also described: basic shapes, evolutions, advantages and drawbacks, and associated components.

Integrating the air intake into the vehicle structure is then considered. Following a few general remarks about the positioning of air intakes in a non-uniform flow, the influence of the wings and fuselage are considered. Examples are analysed for commercial aircraft, fighters, missiles, and possible future air-breathing space launchers.

In the past, the only aspects considered in air intake design were aerodynamic and structural. A few simple aerodynamic computations were made to arrive at a general shape, and this was followed by an extensive wind tunnel testing phase. The complexity of the internal flow, especially in the throat area where this flow is transonic, with multiple interactions between shock waves and boundary layers in the presence of internal boundary layer bleeds, called for a large number of tests because the numerical approach using the efficient Navier-Stokes codes was not possible. Moreover, these tests have to be carried out in large pressurised wind tunnels in order to obtain flows with high Reynolds numbers representative of the whole flight envelope. Some of the set-ups developed for the French ONERA wind tunnels are presented.

Now a days, Radar-Cross-Section is often an essential criterion for military applications, so new design approaches and new locations on the vehicle are being introduced. This problem is presented.

The author concludes this general overview of air intakes with three remarks concerning the air intake specifications, the captured air flow and the internal design.

* Vice President, Research

A Flexible and Automatic Design Environment Applied to the Optimization of Turbomachinery Blades

Stéphane Pierret, Alain Demeulenaere, Bruno Gouverneur
Numeca International,
Av. Franklin Roosevelt 5, 1050 Brussels, Belgium

Prof. Charles Hirsch
Vrije Universiteit Brussels,
Dept. Fluid Mechanics, Pleinlaan 2, 1050 Brussels, Belgium

Abstract

Designing turbomachinery blades is a complex task involving many different objectives and constraints coming from various disciplines. Further improvement of the existing design cycle is probably one of the main challenges of the next decade in the turbomachinery community. Major improvements are expected in terms of reduced design time, reduced engineering time, better optimum and increased design complexity. This challenge can only be tackled by selecting and further developing general and efficient design algorithms integrated into software dedicated to this specific design task.

The most important choice for the success of such a design environment is probably the choice of the design algorithm. Design methods can be grouped into optimization methods, inverse methods and heuristic methods. Their respective advantages and drawbacks are summarized. In the context of a general optimization method applicable to the design of turbomachinery blades, the optimization based on the concept of function approximation seems to have major advantages. A completely new commercial package (FINE™/Design3D) has been developed at NUMECA International that offers more flexibility, improved performance, graphical-user interface (GUI) and full automatization of the design cycle. This new software incorporates various very popular and efficient techniques such as artificial neural networks, genetic algorithms, databases and CFD analysis tools [12]. FINE™/Design3D also includes a very flexible parametric geometry modeler for turbomachinery blades, AutoBlade. AutoBlade is able to represent various turbomachinery blades such as axial or radial compressors and turbines, pumps, blowers, fans, inducers, return channels and turbochargers.

The paper also presents two optimization results obtained with FINE™/Design3D. The first presents the optimization of the stacking law of an axial turbine blade. The efficiency rises by 0.4 % with respect to the original radially stacked blade. The second example presents the optimization of a transonic compressor rotor. The efficiency has been improved from 84.8% to 85.7% after only three optimization cycles.

ISOABE 2001-1060

Low Pressure Turbines With High Stage Loading

R. Vazquez , D. Cadrecha, D. Torre & M. Morales
Aerodynamic Technology Department
Industria de Turbopropulsores, S.A
Madrid, Spain
raul.vazquez@itp.es

T. Crespo,
Mechanical Engineering Technology Department
Industria de Turbopropulsores, S.A
Madrid, Spain

ABSTRACT

In 1999, ITP (Industria de Turbopropulsores, S.A) launched a wide on-going research and development program to advance the state of the art in Low Pressure Turbines for use in future high bypass ratio aeroengines for long range civil transport. The objectives of the program were very aggressive in order to satisfy the market demands. Enormous cost and weight savings (about 30% off) were required, equaling or even improving the efficiency and noise emissions. Through that program, ITP has continuously reduced the components of the turbine, as the most adequate way to achieve the previous targets. As result of that reduction, High Loaded Turbine technology is being developed. High loaded turbines have less number of stages to produce the same work output (high stage loading) and fewer numbers of blades to perform the same duty (high and ultra high lift blades). Although this paper is mainly focusing in the first concept, the second one will also be briefly discussed. An overview of the two types of high loaded turbines, high through flow and low through flow, is given and how both approaches allow an important reduction in weight and number of components (stages and blades). It is also shown that keeping the same levels of efficiency and noise is a technical challenge that demands improved aerodynamics or/and high rotational velocity. To conclude, this paper describes the current technology limits that are exceeded by high loaded turbines and the special aerodynamic and geometrical features that come up.

ISABE-2001-1061

Advanced Low Pressure Turbine Design for A High By-Pass Ratio Aero Engine

Paloma González, Iñaki Ulizar
Aerodynamic technology
ITP, Industria de Turbo Propulsores
Madrid, Spain
Paloma.gonzalez@itp.es , joseignacio.ulizar@itp.es

Victor Olabarria, Josean. Malumbres
Mechanical Design
ITP, Industria de Turbo Propulsores
Bilbao, Spain.
Victor.olabarria@itp.es , josean.malumbres@itp.es

ABSTRACT

In 1997 ITP got involved in the Trent 500 low pressure turbine design joining Rolls Royce Turbine Engineering team and participating not only in the mechanical design but also in the aerodynamic design, analysis and development of the low pressure turbine. The intention of this paper is to give an overview of some of the most relevant characteristics of the LP turbine design and how it was influenced by the experience from previous engines, by research experimental results in the aerodynamic field and also by the introduction of new advanced features, driven by project demands, such as the spoon aerofoils. Although the paper tends to integrate both, mechanical and aerodynamic design, it is mainly focused on aerodynamics.

Simulation of HBPR turbofans with hydrogen fuel

P. Hendrick & K. Van Heste
Royal Military Academy
Applied Mechanics Dept.
Avenue de la Renaissance, 30
1000 Brussels – Belgium
hendrick@mapp.rma.ac.be

ABSTRACT

The performance of classical High By-Pass Ratio (HBPR) and even Very HBPR (VHBPR) turbofans burning hydrogen as a fuel will be estimated through simulation work and thermodynamic cycle calculations.

A potential application of HBPR turbofans burning hydrogen is also presented : a subsonic air-launched vehicle. A part of the launcher is an orbiter which is in fact an all-rocket SSTO launcher taking off from about 10 km altitude and 0.75 - 0.80 Mach.

This type of concept, comparable to the UK Interim HOTOL and the Russian MAKS, is here considered with the use of In-Flight LOX Collection and a ventrally-launched expendable orbiter.

The problems posed by the storage of liquid hydrogen on board the aircraft but also in the airports and by the production of liquid hydrogen in very large quantities will also be treated.

Finally, non classical turbofan configurations will also be proposed. They take larger advantages from the specific characteristics of hydrogen (as its large cooling capacity and low storage temperature).

Low-Cost Free-Flight Testing Of Hypersonic Airbreathing Engines

Jason S. Tyll, Robert J. Bakos
GASL, Inc.
Ronkonkoma, NY 11779

Abstract

Flight-testing is by far the most costly step in the design process for a hypersonic vehicle, which progresses from paper studies, to computer models, to wind tunnel tests, and eventually to full-scale flight. However, it is the essential step needed to demonstrate integration and operability of all flight systems as well as structural durability. Use of an aeroballistics range is being examined as a low-cost alternative to flight testing for scramjet-integrated hypersonic vehicles. The technical feasibility and economical advantages of attaining some portion of the needed flight data will be demonstrated as part of this program. The selected test condition corresponds to Mach 8 flight of full-scale missile at 1000psf flight dynamic pressure. A subscale axisymmetric scramjet-integrated projectile was designed and launched at the AEDC Range-G facility. Additional launches are planned to fully evaluate the test technique. This paper gives a general program overview and discusses the main technical issues associated with the effort. Prelaunch testing in the NASA HYPULSE facility at GASL provided a comparative performance database for the scramjet-integrated projectile. The launch of the first projectile, the structural demonstrator, indicated problems associated with the connectors that attach the cowl to the body and with the sabot design. Both parts failed under launch loads and were redesigned for the second launch, scheduled for mid-June 2001. The ultimate program goal of the current program is to demonstrate supersonic combustion ramjet (scram) operation in a ballistic range and measure powered acceleration using the available instrumentation and measurement techniques. The use of a ballistic range test technique for scramjet flight data can reduce cost by two orders-of-magnitude, from tens of millions of dollars to less than $100K per flight. However, there are also limitations on the type and volume of data that can be acquired due to the combination of scale and launch loads.

Numerical Simulation Of Late-Lean Ignition Processes

by

V. Tangirala, J. Haynes and S. M. Correa
General Electric, Corporate Research and Development
Schenectady, NY 12309

and

R. Seiser
Department of Mechanical and Aerospace Engineering
University of California at San Diego
La Jolla, California 92093

Abstract

Dry low NO_x combustion systems have made it possible to reduce NO_x emissions from gas turbine combustors to below 25 ppm (by vol., dry @ 15% O_2). Further reduction is possible by axial staging of the combustion process in which a stream of combustion products from a lean premixed combustor is used as vitiated air and burnt with fuel injected from a secondary fuel injector. In the present study, numerical simulations are performed using a reactor network model, an eddy dissipation model, a PPDF model and a laminar flamelet model. Laminar flamelet model predictions of exit temperatures of a test-cell combustor, in which fuel is axially staged, show satisfactory agreement with test measurements where as the model underpredicts the NOx concentrations.

Key Words: Combustion, Emissions, Laminar Flamelet Model, Late-lean Ignition

Numerical Simulation of Leading Edge Separation Bubbles on Turbomachinery Blades

by

K.S.Santhosh*, B.V.S.S.S.Prasad and M.Govardhan****
Thermal Turbomachines Laboratory,
Department of Mechanical Engineering
Indian Institute of Technology Madras, Chennai 600036, India

EXTENDED ABSTRACT

This paper deals with numerical simulation of the laminar separation bubbles using finite element method. The code is developed in-house for turbomachinery cascade flows and is based on Taylor-Galerkin formulation. Numerical simulation of flow through turbomachinery passages involves the solution of non-linear partial differential equations expressing the conservation of mass, momentum and energy. In the present analysis, two dimensional, viscous compressible flow equations in conjunction with the equation of state of the fluid are considered. Solution is obtained using explicit time marching method after lumping the mass matrix. An artificial dissipation term is added to keep the solution stable in regions where the gradients are sharp. The amount of dissipation is determined by a pressure based switch in an unstructured mesh environment. A stretched boundary layer grid is provided near the solid boundaries using an exponential stretching function.

The code is validated against standard problems like flow past flat plate, NACA 0012, and a low pressure linear turbine cascade. Apart from validation flow field near two leading edge shapes viz. circular and elliptic fitted to a flat plate is studied. The formation of small laminar separation bubbles and their reattachment near the leading edge is predicted. Elliptic leading edge shape is capable of avoiding separation at zero incidences. Comparisons of C_f and C_p between the two leading edges are given. Flow through a linear turbine cascade (Hodson et al.) with a circular leading edge is simulated at its design conditions. The formation of laminar separation bubbles is observed at off design conditions. In all the cases studied the bubble is found to reattach the blade within a few percentage of chord.

* Research Scholar

** Professor

Corresponding Author : BVSSS Prasad (prasad@iitm.ac.in)

Separation Control in Ultra-high Lift Aerofoils by Unsteadiness and Surface Roughness

O.N.Ramesh[1], H.P. Hodson[2] and N.W.Harvey[3]

ABSTRACT

The high-lift philosophy (Howell at al. 2000) exploits the unsteady wakes which are shed from upstream blade rows to impact on the turbine blade under consideration and initiate laminar-turbulent transition of the suction surface fluid, thereby resulting in a total pressure loss reduction. This enables far higher levels of loading with unsteady flow than was previously possible in the traditional LP turbine bladings with a steady inflow. However, the efficacy of unsteady effects alone to achieve loss reduction seems to reach saturation as the blade loading tends to ultra-high values. This is because such ultra high lift profiles have extremely strong deceleration zones on the aft part of the suction surface and the ill effects of the resulting large separation bubble (and sometimes even open separation) could be only partially contained by the unsteady effects. Hence the aim of this study is to explore if further loss reduction is possible for such ultra-high lift profiles by supplementing the unsteady effects by selective roughening of the aerofoil surface. It is shown in this experimental study that the combination of roughness and unsteadiness can indeed bring about a substantial reduction in total pressure loss. Measurements of surface static pressure distribution and wake traverse of total pressure are shown here which clearly show the substantial loss reduction due to the selective roughness patch in conjunction with the unsteady wakes. Hotfilm anemometry measurements of wall shear stress are analysed to arrive at a mechanism of loss reduction.

1. Department of Aerospace Engineering, Indian Institute of Science,Bangalore, India
2. Whittle Laboratory, Cambridge University, U.K
3. Turbine Engineering, Rolls Royce plc., Derby, U.K.

A Comparative Computational Study Of Jet In Crossflow Using K-ε/LES-Based Turbulence Models

By

M. A. Mawid and T. W. Park
Engineering Research and Analysis Company
Wright-Patterson AFB, OH 45433, USA

B. Sekar
Turbine Engine Division
Air Force Research Laboratory
Wright-Patterson AFB, OH 45433, USA

Abstract

A preliminary computational comparative study of a jet in a crossflow using three different turbulence models was carried out in an effort to assess their predictive capabilities for modeling gas turbine combustors primary and secondary dilution holes jets. The standard two-equation k-e, and LES with Smagorinsky and one-equation k-*l* for the subgrid stress (SGS) tensor were used. The numerical model was based on an all purpose STAR-CD code [24]. A round hole with a crossflow channel and a plenum was utilized as a vehicle to investigate a three-dimensional jet in a crossflow. As a first step, the numerical results were compared with the experimental data of Thole et al. [22] for the plenum case. The comparisons showed that the two-equation k-e turbulence model overall underpredicted the axial velocity component and the peak value of the normal velocity component at stations downstream of the hole exit center. The LES models, however, with both Smagorinsky and k-*l* SGS models overall overpredicted the axial velocity component and also failed to predict the normal velocity component peak downstream of the hole exit center. The jet penetration at the hole exit center was, however, satisfactorily predicted by the three models. The inability of LES to predict the measured velocity field is believed to be due to the models coefficients (i.e., C_s in Eq. (6) and C_v and Ce in Eq. (9)) for both Smagorinsky and k-*l* subgrid scale (SGS) models. Tuning of the models coefficients such that no undershoot or overshoot in turbulence dissipation occurs, is therefore needed.

Key Words: LES, Jet Flow, Turbulence Closures, Crossflow

The Evolution Of The Materials In Gas Turbine Engines/ Influence On The Design And The Performance

Alain.LASALMONIE,
Scientific Adviser
Snecma Moteurs, Rond point G.Ravaud
77550 Moissy Cramayel, France

ABSTRACT

For many years, Aluminium alloys, steels and Ni alloys were predominant in gas turbine engines; they were continuously replaced by new alloys or materials with higher specific strength (Ti alloys, organic matrix composites...) or better temperature capability (single crystals...).

After a description of the past and present situation of the materials in the commercial and military engines, the new material families, which should be introduced in the future engines are briefly described.

The design of modern engines has a strong influence on the material selection and on the manufacturing of the components. New engines have fewer stages, less blades and vanes. It means that the components are highly loaded; moreover, the geometrical shape, which is increasingly 3D, raises new manufacturing problems.

The raise of the temperature in the HP compressor and the turbine , needs new cooling technologies in the metallic components Refractory materials, such as CMC's.; offer new solutions for combustors and nozzles

The impact of environment related concerns is increasingly important for the engine manufacturer .It can deeply influence the design and the processes

New lifing methologies arise from new materials introduction: : probabilistic models had to be introduced to predict the fatigue life of P.M. rotating discs because of the influence of cleanliness on crack initiation. .It is likely that the probabilistic modelling will become increasingly important, when brittle materials such as Intermetallics compounds are introduced. These compounds are indeed highly sensitive to internal or surface defects and scaling effects are expected.

Keywords: Aero engines, high temperature materials, manufacturing, modeling

Computational And Experimental Investigation Of High-Pressure Axial And Centrifugal Compressors With Ultra-High Rotational Speed

Mileshin V.I., Orekhov I., Ogarko N.I., Startsev .N., Pankov S.V.
Central Institute of Aviation Motors (CIAM)
Moscow, RUSSIA

ABSTRACT

The paper presents results of computational and experimental investigation of high pressure axial stage (π_c^* =2.15 U_{tip}=577 m/s) and high pressure centrifugal stage (π_c^* = 8, U2 = 648 m/s, where U2 means rotational speed of impeller trailing edge). Main peculiarity of such type turbomachines is presence of high Mach number regions on tip of rotors due to ultra-high rotational speed. For example, maximum Mach number on tip of axial stage rotor reaches M=2. Tip flow within impellers of high-speed centrifugal compressors is supersonic beginning from leading edge and Mach number on tip of impellers reaches M=1.5÷1.6. The resulting shock waves are very strong and shock loss dominates over other losses. Mentioned circumstances governs the objects of design to achieve acceptable values of adiabatic efficiency. Experimental investigation of axial and centrifugal stages was made at CIAM test rigs. Computation of 3D viscous flow within axial and centrifugal stages was made using Navier-Stokes solver "3D-IMP-MULTI.2000" developed in CIAM. Implicit non-factored modified Godunov numerical scheme (3rd order spatial accuracy) is applied to accelerate convergence process.

ISABE-2001-1124

The Kaveri Engine Technology

Venkataraman Shankar*, T.K Sampath Kumaran and V.Sundararajan+**
Gas Turbine Research Establishment
C.V. Raman Nagar, Bangalore-560093, INDIA

Abstract

January 2001 has been a landmark in the Indian aviation history as the Indian Light Combat Aircraft (LCA) took off to skies. The General Electric (GE) F404 engine powered the flight. Subsequent LCA prototypes are envisaged to fly with the Kaveri engine. Kaveri engine is in the advanced stage of development and in this paper Kaveri engine technology is addressed.

---

* Group Director

** Project Director(Kaveri)

+ Director

Technology Acquisition To The Benefit Of Tm333 Turboshaft Family

By

Patrick MARCONI
Joël SILET
Advanced Project and Performance Department
TURBOMECA - 64511 BORDES CEDEX
France

ABSTRACT

The TM333 turboshaft engine family is intended to power helicopters in the 5 to 6 ton class.

Even if the serial production of the basic version TM333 – 1, certified in 1986 for Dauphin / Panther helicopters, has never been materialised, its development was a technical revolution for TURBOMECA in the 80's. Indeed new components issued from extensive research programmes have been introduced in this new turboshaft engine, and then applied to the new generation of Turbomeca engines.

Then further technological acquisitions have enabled Turbomeca to develop the engine from the basic TM333 –1 (625 kW) to the TM333 –2B2 version offering a 801 kW take–off power for ALH Indian helicopter.

Nowadays, Turbomeca is working on a 895 kW new version, named TM333 2C2, by implementing the results of the latest research and technology programmes.

ISABE-2001-1133

A Physics-Based Approach for the Detection of Cracks in Rotating Disks

Héctor M. Rodríguez and Darren Hallman
GE Corporate Research and Development Center
Niskayuna, New York 12309 USA

Mark Butz
GE Aircraft Engines
Cincinnati, Ohio 45215 USA

Vikas Dhau
GE Research and Development Center
Bangalore, India

ABSTRACT

A physics-based approach for the detection of cracks in rotating disks is presented. The approach takes advantage of the distinct behavior in the vibration response of cracked rotating disks. As shown in the manuscript, the crack opening yields an additional unbalance that is proportional to the square value of the speed, and hence the resulting crack-induced unbalance force is proportional to the fourth power of the speed. This unique unbalance response characteristic brings the opportunity to implement an on-line monitoring system to detect early stage disk cracks by measuring the vibration response of the rotating assembly at non-invasive locations.

Evolution Of Nonuniform Radial And Tangential Temperature Fields In Multistage Turbines

A.V. Granovski, M.Ja. Ivanov, R.Z. Nigmatullin, L.V. Terentieva
Central Institute of Aviation Motors (CIAM) 2,
Aviamotornaya St., Moscow, 111250, Russia

Abstract

In the present work the propagation of inlet total parameter distributions through multistage turbine flow passage is studied using different approaches. One approach uses a development of S_2-surface methods based on Euler and Navier-Stokes equations. In the other approach the gas flow passage simulation uses 3D Euler equations with additional terms for prediction of evolution effects. The last method is based on 3D Navier-Stokes equations. Detailed comparisons between computational and experimental data are presented.

Key words: multistage turbine, temperature distribution

ISABE-2001-1165

Performance Of Modern Stovl Fighter Powerplants

Roy A. Cookson, Jose Luis Herrero, Dimitris Myzithras and Pericles Pilidis
School of Engineering, Cranfield University
Cranfield, Bedford, U.K

Abstract

The aim of this paper is the description of the performance characteristics of powerplants intended for modern STOVL fighters. The intended application for the propulsion system investigated here is a combat ASTOVL aircraft. Two powerplants are examined, as they appear to be the most viable solutions for a combination of high performance, increased efficiency and STOVL capability. These powerplants are the lift-nozzle and the low pressure shaft driven lift fan. Both powerplants involve the integration of a similar core, with different approaches for providing vertical thrust. They appear to be viable solutions for a combination of high performance, good efficiency and STOVL capability.

One, the engine with lift nozzles, was composed of a gas generator with an overall pressure ratio of 28, BPR 0.6, TET 2050K at cruise conditions of 1.2 Mach number and 30,000ft. The other, the shaft driven lift fan, was a dual cycle powerplant. It was composed of a conventional cruise engine (OPR 28, TET 2050K and BPR 0.5). The core was optimised for forward cruise flight at the same conditions as the previous one. For ASTOVL, a shaft driven remote lift fan, is used to generate vertical thrust.

The net thrust needed to sustain supersonic cruise at the design point was 65kN. The higher bypass core engine of the vectored thrust engine gives it a small SFC advantage, although this advantage appears to be smaller at high flight speeds. Also, the net thrust produced by the core with the lift nozzles is slightly higher in all the off-design situations, showing 130kN against those 128kN of the lift fan configuration at SLS conditions, or the 192kN against 185kN, relatively, for augmentation at design point conditions.

Once the cruise engine performance prediction was made, the STOVL simulation was attempted. The shaft driven lift fan overall powerplant's cycle was changed, as the lift generation system acts when power is extracted from the cruise engine, to offer a total vertical thrust of 164kN, against 130kN from the vectoring lift thrust engine. This shows a clear advantage for STOVL operations, backed by the better tolerance of that powerplant to the inlet temperature increase, which simulates the hot gas ingestion. However this engine is expected to be somewhat heavier.

The performance simulation described here highlights many salient features of the engine. The scope of this study was to compare the basic performance of these novel powerplants. From these further assessments can be carried out in the future related to a more detailed understanding of these advanced powerplants.

ISABE-2001-1178

Coke Deposit Formation At Aviation Gas Turbine Engines: Problem Of Coke Formation And Supression And Removal

By

Yanovskiy L.S., Ivanov V.F., Baikov A.V.
Central Institute of Aviation Motors, Moscow, Russia

Abstract

The data of new researches in area of enhancement of thermal oxidative stability (TOS) of jet fuels are submitted. The results of tests of CIAM developed multifunctional additive for jet fuels are presented. This additive contents an antioxidizer, metal deactivator and dispersant and one (in concentration 0.05%) raises TOS of jet fuels TS-1 and Jet A-1 by 100 - 120 F that allows essentially to lower the gum and coke formation. The results of research of coke deposit formation into manifolds of gas turbine engine (GTE) combustors and afterburners at stationary and transitive modes of operations are submitted.

It is shown that the coke deposit formation at the fuel flow in manifolds and atomizers of GTE is determined in general by the kinetics of thermal oxidation of fuels and admixtures into volume and by the diffusion transfer of the reaction product to the walls and by the wall catalytic heterogeneous reactions kinetics. In this case the various combinations of hydrodynamic and heat modes of GTE operation, and of the properties of fuel and wall material result in various and to apriori unobvious tendencies of coke deposit formation in manifolds. The radical-chain chemical mechanism of the thermal oxidation with degenerated branching of chains is proposed by authors for the description of the coke formation. The various models of kinetics of coke deposit formation are proposed. Application of these models in unit of the equations of hydrodynamics, heat- and mass transfer for fuels and the reaction products allowed reliably to determine the coke deposit formation into manifolds and atomizers of GTE at the fuel heating in real conditions. The results of the modeling of the coking intensity of the GTE fuel manifolds and atomizers at the stationary modes as well as at the unstationary modes of the operation (start-up, transitional mode and stop) are presented. There are shown that core deposits are formed most intensively at the end cross-sections of manifold and end sprayers of GTE main combustion chambers.

For GTE afterburners there are shown that coke deposits are formed in general at unstationary modes, particularly, at the start when fuel is fed of manifold heated up to hot gas temperatures. During manifold cooling the coke formation are stopped at the first cross-sections of manifold and inhibited strongly at the end cross-sections: coke deposits "wave is running" into manifold at the start. Coke is formed into all parts of manifold and into all atomizers.

The effects of different constructional factors (additional heat protection of sprayers corpuses, heat transfer intensification by means of microribbing of fuel channels) are determined. Other new effects are revealed also. These effects were verified by the test results.

The developed methods and the codes for the coke deposit formation allow to optimize the GTE manifolds and atomizers. These constructions maintain the minimization (suppression) of coke deposits. The effective method of the removal of the coke deposits from the cavities of pipelines maintaining the complete refining of GTE fuel channels is developed. This method is based on fuel channels with coke deposit treatment by gas blend of special composition at low temperatures (20-60^{0}C) and next wash of fuel channels by the organic and inorganic water solutions.

ISABE-2001-1182

Gas-Particle Flows In Co_2 Pellet-Blasting Nozzles

A. Hamed* and O. Mesalhy †
Department of Aerospace Engineering & Engineering Mechanics
University of Cincinnati, Cincinnati, Ohio

T. Lehnig ‡
Vice President and Director of Engineering and Operations
Coldjet, Cincinnati, Ohio

ABSTRACT

A numerical study was performed to investigate gas-particle flows in convergent divergent CO_2 pellet-blasting nozzles. The solutions to the compressible Navier-Stokes equations and the equations governing the particles' motion through the gas, in a Lagrangian frame of reference are solved iteratively in order to model the two-way mass momentum and energy exchanges between the two phases. Results are presented for the gas flow field and particle velocities at different particle loadings and inlet velocities. The momentum and energy exchange with the particles reduce the gas velocity in the divergent passage near the center. The presented results demonstrate the effects of particle loading and inlet velocity ratios on pellet and gas velocity and temperature through the nozzle.

* Professor, FelloAIAA

† Doctoral student

‡ Member AIAA

ISABE-2001-1186

Experimental Analysis Of Thrust Vector Control Using External Vanes

W.A. Aissa[*], A. A. Hashem[**]

Abstract

Thrust vector control (TVC) can be used to augment or replace aerodynamic control forces. It is the only option for post-stall (PS) maneuverability. A static test is conducted to investigate the internal performance of axisymmetric convergent sonic nozzles under the effect of vectoring two externally mounted vanes. The tests cover typical military and maximum nozzle throat areas, at different nozzle pressure ratios and vane deflection angles. The tests have been conducted in a transient mode; however, it has been proven that the results can be treated as quasi-steady. The nozzle/vane performance parameters are obtained and arranged using newly defined force and moment coefficients to generalize test results and help design similar systems.

Key words : Thrust vector control, vane type, aerodynamic performance.

[*] High Institute of Energy, Aswan., Mech. Power Dept. : walid_aniss@yahoo.com

[**] Cairo University, Faculty of Engineering, Aerospace Department. : aliahashim@hotmail.com

ISABE 2001-1192

Mach 10 Scramjet Design and Ground Test Challenges

Dr. John I. Erdos, Dr. Robert J. Bakos, Dr. Anthony Castrogiovanni
GASL, Inc.
(an Allied Aerospace Industries, Inc. Company)
Ronkonkoma, NY, USA

ABSTRACT

The problems in designing and ground testing airbreathing engines capable of use on future space launch systems are examined. The rationale for use of airbreathing engines on space launch systems, or hypersonic cruise vehicles, is briefly reviewed to place their problems in context. The design challenges that are addressed here are structural, and result mainly from the transient thermal loads imposed by acceleration in the hypersonic flight regime. The implications of local "hot spots" associated with shock impingement on or near the cowl leading edges and with internal shock impingement and reflections throughout the combustor (which are unique to the hypersonic regime) are discussed in relation to the design of fuel-cooled structures. Demonstration of engine structural integrity, as well as performance and operability, will necessarily be completed through flight tests. However, to the extent that ground testing is feasible, flight test risk and cost will be reduced. Therefore, the challenges in performing meaningful ground tests of hypersonic airbreathing engines are also discussed. In view of the fact that, at the present, only very short duration tests in pulse-type tunnels, and in aeroballistic ranges, are feasible at Mach 10 and higher, some history and experience relevant to the evident issues associated with such tests are reviewed. Facility and instrumentation capabilities, and approaches to model design, are discussed.

Influence Of A Rotor-Stator Interaction on The Steady and Unsteady Characteristics of The Axial Compressor

V.E. Saren, N.M. Savin, V.G. Krupa, A.M. Petrovitchev
CENTRAL INSTITUTE OF AVIATION MOTORS (CIAM)
2, Aviamotornaja st., Moscow, 111250, Russia

ABSTRACT

The results of experimental and theoretical researches of unsteady (periodic) flow in a system of blade rows stator-rotor-stator of the axial compressor are presented. The main attention is given to the gasdynamic mechanism of the stators clocking influence on pressure fluctuations, pressure ratio and efficiency of the compressor.

It is shown that the stators clocking effect is aroused by dissipation of the unsteady vortexes behind rotor blades.

Key words: Turbomachinery, Unsteadiness, Clocking effect.

Mathematical Modeling Of Flow In A Wave Rotor and Its Comparison with Experimental Data

A.N. Kraiko, N.I. Tillyayeva
CIAM, RUSSIA

Abstract

Numerical modeling of flow in a wave rotor - WR passage is designed. A base of this mathematical model is one-dimensional unsteady Euler equations system. Some non-one-dimensional and non-ideal flow effects are taken into consideration. These are change of flow parameters due to gas inflowing into wave rotor passages - WRP from inward ports and outflowing through outward ones, not instantaneous opening and closing of WR ends - windows, friction of gas about walls of WRP, leakage of gas into inner volume of WR and backward through the gap between ends of its passages and casing. For nine three-port WR, that were experimentally investigated in Lewis Center (NASA) and Power Jet Corporation, the comparison of those experimental characteristics and obtained at the base of developed model computational ones is conducted.

Experimental Investigation Of Front Devices With Double Swirlers For Perspective Combustors

Rutovskiy V.B., Kravchenko I.V.
Moscow State Aviation Institute (Technical University)
Volokolamskoe shosse, 4, Moscow, A-80, 125993, Russia

ABSTRACT

The purpose of the presented investigation is to obtain deeper understanding of work process of burners with swirlers necessary for design of combustors for perspective gas turbine engines with high combustion efficiency and low emission level.

The paper includes the results of experimental investigation of the physical basis of the working process in the swirl burners.

It seems advisable to conduct the investigation of burner parameters impact on flame stabilization process using burners placed in free space. In this case the conditions in the space surrounding the burner would be known unlike the version with burner mounted in combustion chamber. This scheme allows carrying out the investigation of the principles of swirl burner working process and of the influence on this process of most significant operation and structural parameters. Essential problem is the study of flow structure after the burner, since it has decisive influence on flame stabilization process. The elaborating of physical model of flame stabilization in swirl burner allows to determine the influence on this process of principal design parameters of burner in itself and to establish the rational range of their change.

Five types of burners investigated. To determine the common regularities of flow, the structure in the cross-sections of a jet with out flame was determined. Except a research of a structure for of cold pressurization's, the measurements of fields of temperatures behind burners with flame are conducted

On the basis of studies of flow structure and burning process carried out, one may schematically present the mechanism of flame stabilization in the investigated burners.

It has been demonstrated that such a swirl burner generates zones of a strong rotational shift and high turbulence level, decreases axial component of the velocity, causes axial velocity reversing near the exit, which leads to generation of a recirculation zone, i.e. a zone of intense mixing with combustion products, as well as produces a tangential component of the flow velocity.

The performed study makes it possible to elaborate a correct approach for designing and utilization of swirl burners. Comparison of different operation modes of these swirlers allows giving some well-defined recommendations.

In short combustors of aviation engines, for quick destruction of the swirling airflow structure and provision of uniformity of the parameters at a combustor's exit, reverse-flow swirlers should be used.

ISABE-2001-1221

A Compliant Casing For Transonic Compressors

Gregory S. Bloch
Propulsion Directorate, Air Force Research Laboratory
Wright-Patterson AFB, OH

Chunill Hah
NASA Glenn Research Center
Cleveland, OH

Abstract

It is well-known that a small tip clearance is desirable to minimize the tip-leakage flow and its associated losses in axial-flow compressors. If a casing rub event occurs, however, both the casing and the rotor will be damaged and the clearance between the rotor tip and the casing will be permanently increased. This increase in tip clearance will result in a permanent decrease in aerodynamic performance until the gas turbine engine is removed from service and the damage caused by the casing rub event is repaired. This paper presents an application of brush seal technology to create a compliant casing for a transonic axial-flow compressor. This compliant casing has demonstrated the ability to withstand rub events without sustaining damage to either the casing or rotor and without any degradation of aerodynamic performance subsequent to the rub events. This rub-tolerant casing produced the same pressure rise and efficiency characteristics as the original casing, but with an improvement in mass flow range.

ISABE-2001-1226

Evolution of Combustion Technology For Small Aviation Gas Turbines

P. Sampath
T.C.J. Hu
H. Shafique
Pratt & Whitney Canada Corp.
1801 Courtney Park Drive
Mississauga, Ontario
L5T 1J3 Canada

ABSTRACT

The market place for small aviation gas turbines continues to evolve at a brisk pace. This has resulted in paradigm shifts in engine design philosophies. Combustion technology and design methods have evolved to meet the needs of the market place, which include improvements to fuel burn, specific power, reliability, emissions and ownership costs. Process improvements in design, development and manufacturing of combustion system components have been aimed at reducing leadtimes, improving product quality and meeting customer requirements. Advancements in combustion technology aligned with improved numerical tools and manufacturing methods are discussed, along with comparisons to more traditional methodologies. Some future challenges are also discussed.

ISABE 2001-1248

System Studies Of Marinization Of Aero Gas Turbines

Saptarshi. .M* and J.S.Rao**
***BE Marine Engineering Research Institute, Kolkatta**
and **Bharat Heavy Electrical Limited, Hyderabad : 500 093, India

Abstract

Marinization of an Aero Gas Turbine (GT) in particular making it suitable for warship propulsion involves the redesign of the jet engine to make it suitable for providing power in a corrosive environment at sea level, while burning Dieso/Heavy oil. Stringent noise and shock requirement must be met. Intake air and turbine exhaust has to be ducted to and from the GT over a considerable distance and the controls and power output must be integrated to the ships system. The paper describes the typical problems involved and development required when making an aero GT suitable for warship propulsion application and the various system integration requirements of lube oil, cooling air system, engine control and starter system. The major factor to be borne in mind when marinizing an aero gas turbine is modifying critical components in order to increase the time between overhauls and improved O&M procedure.

* BE - MERI, Kalkotta

** DGM, BHEL, Hyderabad

ISOABE 2001-1060

Low Pressure Turbines With High Stage Loading

R. Vazquez , D. Cadrecha, D. Torre & M. Morales
Aerodynamic Technology Department
Industria de Turbopropulsores, S.A
Madrid, Spain
raul.vazquez@itp.es

T. Crespo,
Mechanical Engineering Technology Department
Industria de Turbopropulsores, S.A
Madrid, Spain

ABSTRACT

In 1999, ITP (Industria de Turbopropulsores, S.A) launched a wide on-going research and development program to advance the state of the art in Low Pressure Turbines for use in future high bypass ratio aeroengines for long range civil transport. The objectives of the program were very aggressive in order to satisfy the market demands. Enormous cost and weight savings (about 30% off) were required, equaling or even improving the efficiency and noise emissions. Through that program, ITP has continuously reduced the components of the turbine, as the most adequate way to achieve the previous targets. As result of that reduction, High Loaded Turbine technology is being developed. High loaded turbines have less number of stages to produce the same work output (high stage loading) and fewer numbers of blades to perform the same duty (high and ultra high lift blades). Although this paper is mainly focusing in the first concept, the second one will also be briefly discussed. An overview of the two types of high loaded turbines, high through flow and low through flow, is given and how both approaches allow an important reduction in weight and number of components (stages and blades). It is also shown that keeping the same levels of efficiency and noise is a technical challenge that demands improved aerodynamics or/and high rotational velocity. To conclude, this paper describes the current technology limits that are exceeded by high loaded turbines and the special aerodynamic and geometrical features that come up.